Proceedings of the ASME 2019 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems
InterPACK2019
October 7-9, 2019, Anaheim, California, USA

IPACK2019-6381

EVALUATION OF LOW ORDER STRESS MODELS FOR USE IN CO-DESIGN ANALYSIS OF ELECTRONICS PACKAGING

L. M. Boteler[1], S. M. Miner[2]

[1]U.S. Army Research Laboratory, Adelphi, MD
[2]U.S. Naval Academy, Annapolis, MD

ABSTRACT

Co-design and co-engineering have the potential to improve the design of electronics packaging significantly. A co-designed approach moves away from the sequential approach of an electrical layout followed by a mechanical module design, and then the addition of a heat sink. Replacing it with an approach that addresses the electrical, thermal, and mechanical design simultaneously during the initial design. The goal is to evaluate the design space quickly, considering both the thermal and mechanical stress aspects together. ParaPower is a low order fast running parametric analysis tool, developed by the Army Research Laboratory (ARL), that allows rapid evaluation of package temperatures and coefficient of thermal expansion (CTE) induced stresses throughout the design space. The model uses a 3D nodal network to calculate device temperatures and thermal stresses. In order to rapidly evaluate the design space both the thermal and stress models must be reduced order and provide reasonable results on coarse grids. In the case of the stress model, the goal is a low order relationship between the temperatures and the CTE induced stresses. This paper compares three different low order models for stress. The first uses a more traditional planar module design. This assumes a substantial substrate or heat spreader as the base for the module assembly. The second model is less restrictive, eliminating the requirement for a substrate. The third model also eliminates the substrate requirement, but also allows for in-plane distribution of the stresses. The first two models do not account for the in-plane distribution. Two geometries are considered, a standard power module with a substantial substrate and a stacked novel module with no clear substrate layer. Results for both geometries and the three stress models are compared to finite element analysis (FEA) using SolidWorks, beginning with a thermal analysis followed by a stress analysis based on the temperature solution. All three models run roughly two orders of magnitude faster than the FEA and they correctly predict the trends in the CTE induced stresses.

NOMENCLATURE

E	Young's modulus
F_x	x direction force
F_y	y direction force
L_F	Final length
L_o	Initial length
L_T	Unrestrained expansion/contraction length
T	Temperature
C	Uniform component of strain
r	Radius of curvature
t_b	Bending axis location
z	Vertical location in the substrate/film stack
α	Coefficient of thermal expansion
ε	Strain
ν	Poisson's ratio
σ	Stress

1. INTRODUCTION

In previous work [1-5], the authors described and demonstrated the use of a co-design approach in the ParaPower parametric analysis tool. The parametric analysis combined the thermal and CTE stress analyses providing an evaluation that addressed both aspects simultaneously. The analysis was applied to both a typical planar type power module geometry and a stacked geometry. The planar geometry is shown in figure 1. In the case of the planar geometry, validation against standard FEA results showed that the temperatures were predicted to within 3.5°C and the stresses to within 30%. Details of the analysis are included in [2]. The power dissipation for the chips was $1.25X10^6$ W/m^2, the convection coefficient on the backside of the heat sink was 20,000 W/m^2 °C and the zero stress processing temperature was 217°C.

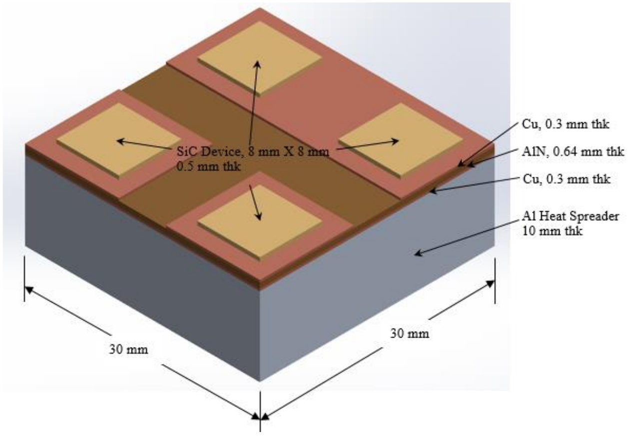


FIGURE 1: PLANAR MODULE GEOMETRY

The stacked geometry core is shown in figure 2. This core is placed inside a copper casing filled with an epoxy encapsulant that provides voltage stand-off. The casing is finned to provide enhanced surface area for convection. The AlN fins are thermally connected to the case and the ends are capped with plastic end caps.

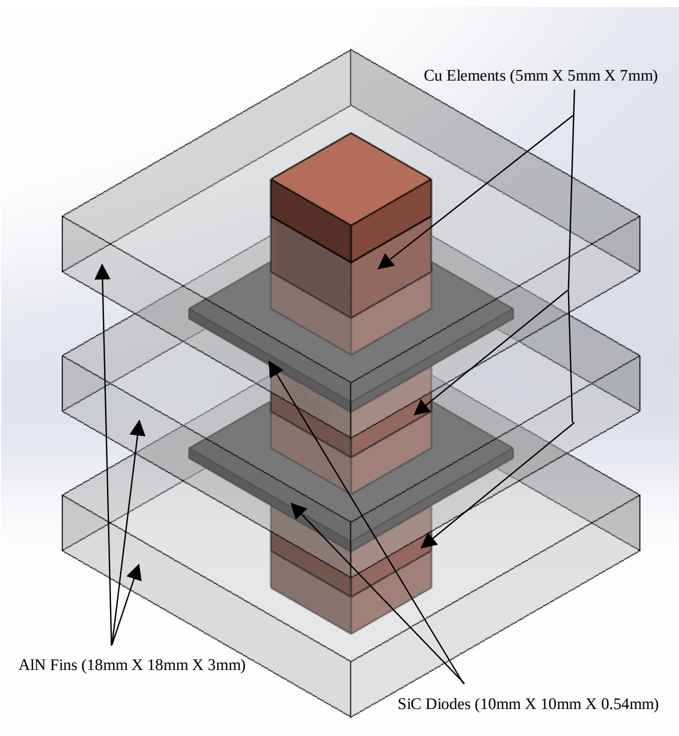


FIGURE 2: STACKED GEOMETRY CORE

In the case of the stacked geometry, validation against standard FEA results showed that the temperatures were predicted to within 3.5ºC but the magnitude of the stresses was significantly over estimated. Details of the analysis are included in [3]. The power dissipation for the chips was $1.6X10^6$ W/m^2, the convection coefficient on the finned surface of the copper casing was 5,000 W/m^2 ºC and the zero stress processing temperature was 217ºC.

The present study evaluates three stress formulations for the analysis of CTE induced thermal stress. The three stress models are a substrate dominated model without intra-planar effects, a non-substrate dominated model without intra-planar effects and a non-substrate dominated model that includes intra-planar effects. The first model was used to evaluate stress in the planar geometry [2]. The first and second models were used to evaluate the stress in the stacked geometry [3]. This paper introduces the use of the third model which is used to evaluate the thermal stresses in both geometries. Thermal stress is calculated for both geometries using all three models and the results are compared to FEA results.

2. GEOMETRY

The grids used for the ParaPower analysis and the FEA for the planar geometry are shown in figures 3 and 4, respectively.

The ParaPower grid models the devices as two layers, a lower layer 0.4 mm thick and an upper layer 0.1 mm thick, where the power dissipation is included. The aluminum heat spreader is divided into two, 5 mm layers. The total grid size is 12 X 12 X 7 for 1008 nodes.

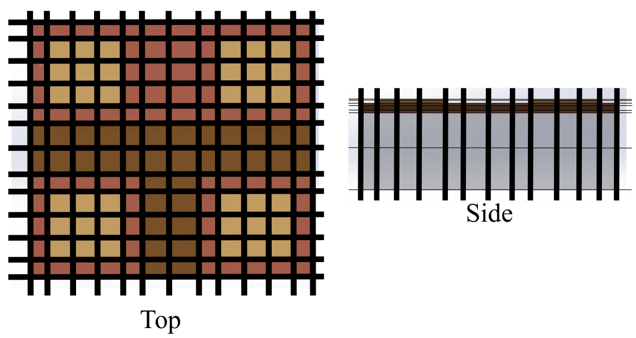


FIGURE 3: PLANAR GEOMETRY PARAPOWER MESH

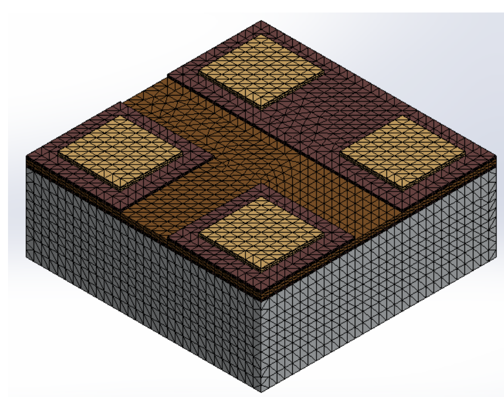
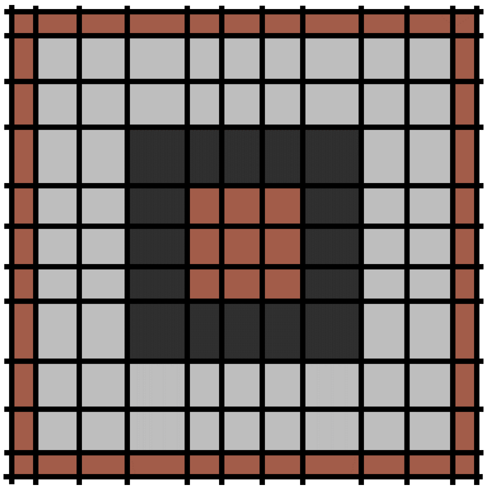


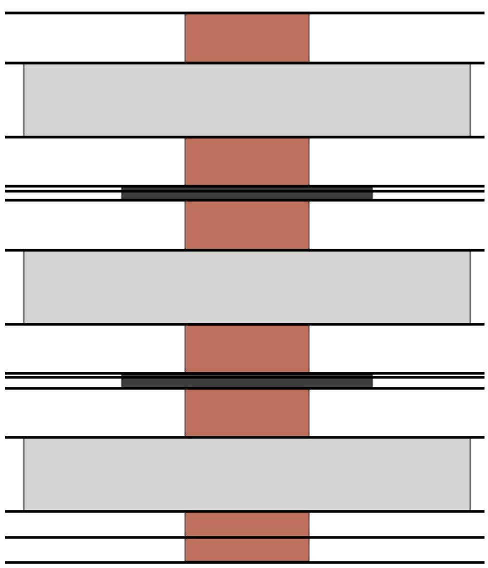
FIGURE 4: PLANAR GEOMETRY FEA MESH

The FEA mesh contains 145,771 elements. SolidWorks was used to create the solid model and the subsequent mesh. In addition, SolidWorks was used to perform both the thermal analysis and the stress analysis.

Figure 5 shows the ParaPower grid for the stacked geometry. Again, the devices are two layers, a lower layer 0.5 mm thick and an upper layer 0.05 mm thick, where the power dissipation is included. The total grid size is 11 X 11 X 14 for 1694 nodes.



Top



Side

FIGURE 5: STACKED GEOMETRY PARAPOWER MESH

Figures 6 and 7 show the FEA mesh, which contains 73,578 elements. Again, both the thermal and stress analysis were performed using SolidWorks.

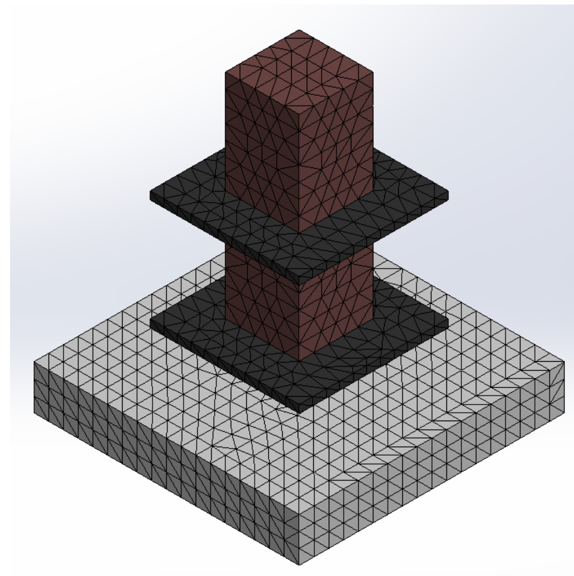


FIGURE 6: CORE MESH, TOP AND CENTER FINS REMOVED TO SHOW DETAIL

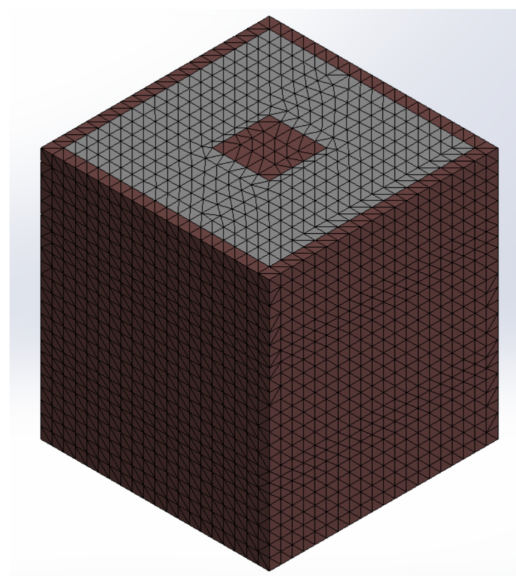


FIGURE 7: OVERALL ASSEMBLY MESH

3. THERMAL STRESS MODEL

The substrate dominated model for thermal stress analysis is based on work by Hsueh [6]. That analysis treats the system as a substrate layer on which the other film layers are stacked. Figure 8 illustrates the elements that are included in calculating the stress at a particular location.

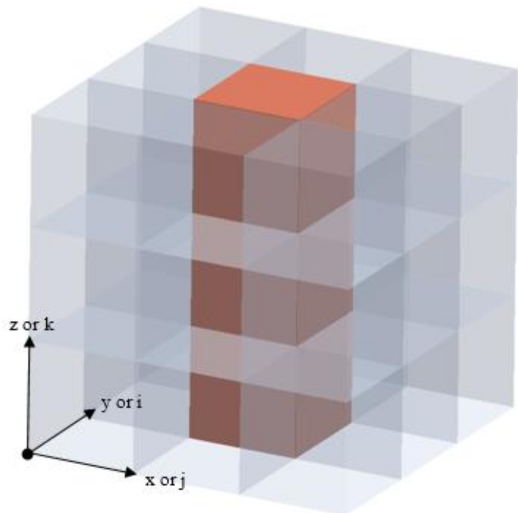


FIGURE 8: SUBSTRATE BASED ANALYSIS ELEMENTS

The equations for the stress in the substrate and the individual film layers are:

$$\sigma_S = E_S\left(\varepsilon - \sigma_S \Delta T\right) \text{ stress in the substrate,}$$
$$\sigma_i = E_i\left(\varepsilon - \sigma_i \Delta T\right) \text{ stress in individual film layers.} \quad (1)$$

In both equations, the value for E is the biaxial modulus given by $E/\left(1-\nu\right)$. This accounts for two dimensional planar stress, as opposed to a single dimension uniaxial stress. At each layer in the assembly, substrate and film(s), the stress distribution is assumed to be planar. The value of ε is given by $\varepsilon = c + \frac{z - t_b}{r}$, where the first term on the right represents the contribution from the uniform strain component and the second term is the contribution from the bending strain. This model considers all layers at a particular i,j location, each i,j location is independent of others. Equilibrium is enforced at each i,j location by forcing the sum of the forces in the stack to be zero. This model gives the stress in x and y directions, but not z. In this paper, the exact formulation proposed by Hsueh is used. This formulation is appropriate for the planar geometry of Figure 1, but not as applicable to the stacked geometry of Figure 2.

An alternate stress model was developed that did not impose the substrate requirement of Hsueh's model [3]. The same elements used in the Hsueh model to calculate the stress are used in this model. In this model, the elements in each layer at a specific i,j location are first allowed to expand or contract (L_{Ti}) based on the difference in temperature between the processing temperature and the operating temperature. Then the strain for that element is based on the difference between that element's length and final length of all the elements (L_F) in the stack of elements at that i,j location. The final length is based on the equilibrium of the forces in all the elements in the stack of elements at that i,j location. Equation (2) outlines the formulation.

$L_{Ti} = L_o(\alpha_i \Delta T_i + 1)$ free expansion length based on CTE

Final length based on restrained expansion

$$F_{ix} = \sigma_i dydz \qquad F_{iy} = \sigma_i dxdz$$
$$\sum F_{ix} = 0 \qquad \sum F_{iy} = 0$$
$$L_{Fx} = \frac{\left(L_o \sum E_i dydz\right)}{\sum \frac{(E_i dydz)}{(\alpha_i \Delta T_i + 1)}} \qquad L_{Fy} = \frac{\left(L_o \sum E_i dxdz\right)}{\sum \frac{(E_i dxdz)}{(\alpha_i \Delta T_i + 1)}} \quad (2)$$

$\varepsilon_i = \left(L_F - L_{Ti}\right)/L_{Ti}$ final strain based on equilibrium

$\sigma_i = \varepsilon_i E_i$ stress for each element in stack

Again, this model gives the stress in x and y directions, but not z. Both of these stress formulations only consider a vertical stack of elements from the bottom layer to the top layer. This accounts for inter-layer differences in stress. They do not account for the intra-layer effects that result from the interaction of different materials in a specific layer.

The third stress model is a quasi 3-D approach that takes a planar approach to determining the final length of an element based on all the elements surrounding it in the same plane. Figure 9 illustrates the elements that are included in calculating the stress at a particular location. This allows the intra-layer effects of different materials in a layer to influence the final length of all the elements in that layer. In this model, all the elements in a layer are first allowed to expand or contract (L_{Ti}) individually based on the difference in temperature between the processing temperature and the operating temperature for that element. Then the strain for that element is based on the difference between that element's length and final length of all the elements (L_F) in that plane of elements. The final length is based on the equilibrium of the forces in all the elements in that plane. Equation (3) outlines the formulation. This model gives the stresses in all three directions due to consideration of the three planes shown in figure 9. Because the planes are considered individually, this analysis is quasi 3-D.

In all three models, only the normal stresses are calculated. In particular, for the substrate based formulation and the first non-substrate based model only σ_x and σ_y are calculated. In the case of the second non-substrate based model, all three normal stresses are calculated. Inclusion of shear stresses would significantly increase the level of complexity and the time required to run the analysis.

4. RESULTS

Comparisons between ParaPower results and FEA results are for the von Mises stress. In the case of the substrate based and the first non-substrate based models, the von Mises stress computation uses σ_x and σ_y. For the second non-substrate based model, all three normal stresses are used.

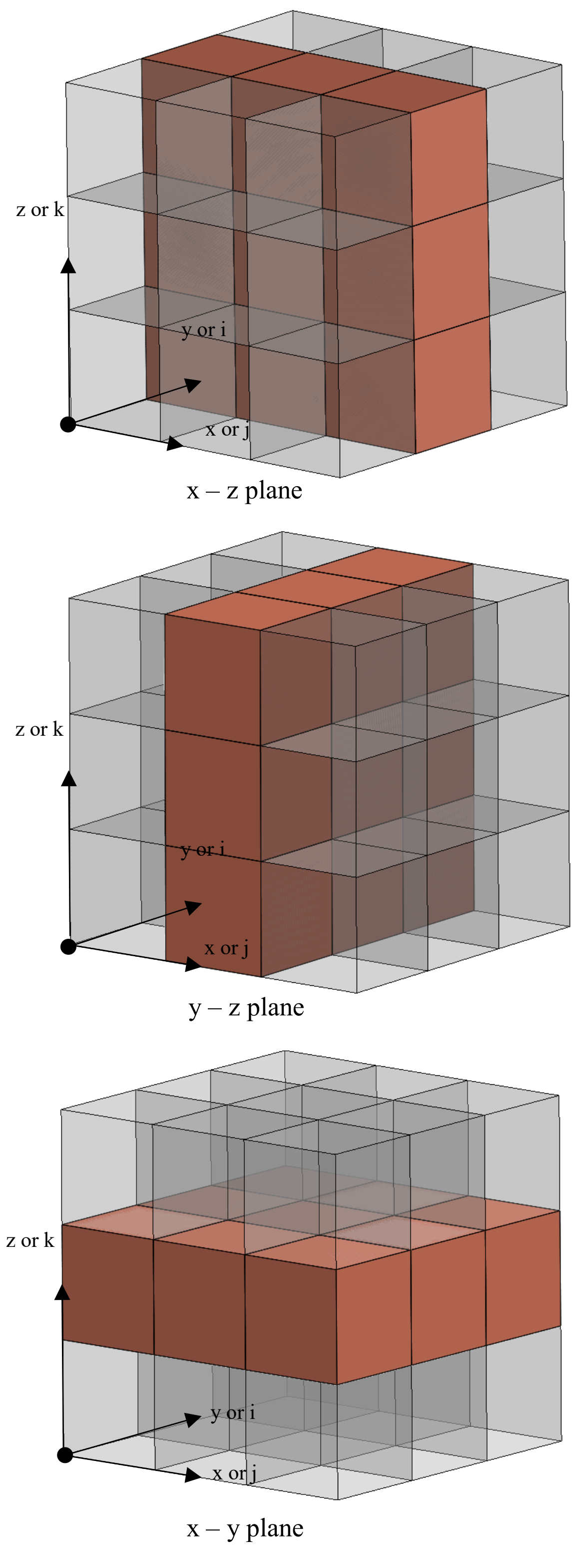


FIGURE 9: QUASI 3-D ANALYSIS ELEMENTS

$L_{Ti} = L_o(\alpha_i \Delta T_i + 1)$ free expansion length, CTE based.

Final length based on restrained expansion:

$$F_x = \sigma_{ik} dydz$$

$$\sum F_x = 0$$

$$L_{Fx} = \frac{\left(L_o \sum_i \sum_k E_{ik} dydz\right)}{\sum_i \sum_k \dfrac{(E_{ik} dydz)}{(\alpha_{ik} \Delta T_{ik} + 1)}}$$

$$F_y = \sigma_{jk} dxdz$$

$$\sum F_y = 0$$

$$L_{Fy} = \frac{\left(L_o \sum_j \sum_k E_{jk} dxdz\right)}{\sum_j \sum_k \dfrac{(E_{jk} dxdz)}{(\alpha_{jk} \Delta T_{jk} + 1)}} \qquad (3)$$

$$F_z = \sigma_{ij} dxdy$$

$$\sum F_z = 0$$

$$L_{Fz} = \frac{\left(L_o \sum_i \sum_j E_{ij} dxdy\right)}{\sum_i \sum_j \dfrac{(E_{ij} dxdy)}{(\alpha_{ij} \Delta T_{ij} + 1)}}$$

$\varepsilon_i = \left(L_F - L_{Ti}\right) / L_{Ti}$ final strain based on equilibrium.

$\sigma_i = \varepsilon_i E_i$ stress for each element in the plane.

4.1 Planar Geometry

Results for the planar geometry are presented for all three stress models for the lower left quadrant. Figure 10 shows this region. The labels for the different stress models are Eqn (1) for the substrate based model, Eqn (2) for the non-substrate based model formulation given in equation (2) and Eqn (3) for the

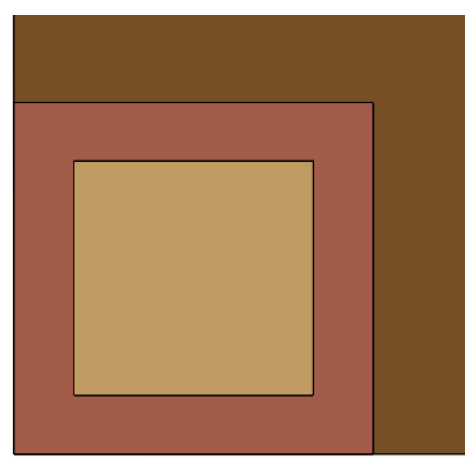


FIGURE 10: RESULTS DISPLAY REGION

quasi 3-D formulation given in equation (3). For figures 11-15 the numerical values from ParaPower are superimposed on the color contour stress results from the FEA. The values from the ParaPower results are printed at the centers of the respective elements.

Figures 11-14 show comparisons for the active surface of the device, the copper layer just below the device, the aluminum nitride layer and the lower copper layer. Note that the models based on Eqns (1), (2) and (3) do not adequately capture the peak stresses that occur at the corners of the geometry, where there are significant shearing stresses. Shearing stresses are not included in any of the models. Inclusion of the shearing stresses would require a coupled set of equations in the x, y and z directions using offset grids. This is contrary to the purpose of ParaPower, which is to evaluate a design space quickly for trends that would drive the design. Therefore, in evaluating errors for the models, comparison is made just away from these peak stress areas, which are primarily corner regions where different materials are attached to each other. Figure 11 shows that all three models predict peak stress at the upper right corner of the device, with the lowest stress in the lower left corner. This trend agrees with the results from the FEA. The FEA shows peak stress of roughly 475 MPA. The Eqn (1), (2) and (3) models have peak stresses of 534 MPA, 1135 MPA and 1225 MPa, respectively. This results in errors of 12%, 139% and 159%, respectively. For the top layer of Cu, figure 12 shows that the Eqn (3) based model does the best job of predicting the trends in the stress distribution, with the peak value in the lower left corner and the minimum in the upper right corner. Errors in the value of the peak stress are 25%, 74% and 75% for the Eqn (1), (2) and (3) models, respectively. Figures 13 and 14 show similar results for the AlN and lower Cu layers, with the Eqn (3) model best representing the distribution of stress in these layers. The respective errors in peak stress for the three models are 14%, 31% and 31% for the AlN layer, and 1%, 56% and 57% for the lower Cu layer. These results show that a substrate based model does a better job of predicting the magnitude of peak thermal stress for geometries that are mounted on a thick substrate layer, but that the quasi 3D model is better for predicting the distribution.

Further evaluation of the substrate based model included running the planar geometry with a finer grid, 30 X 30 X 17, for 15,300 nodes. The purpose of this run was to evaluate grid independence. Figure 15 shows the results for the device layer. Comparison to figure 11 shows that the values and distribution of the stress predicted by ParaPower did not significantly change after doubling the grid size.

The final evaluation was a parametric analysis of the stress as a function of backside convection coefficient. Five values of h were used, 5000, 10000, 20000, 40000 and 100000 W/m^{2}K. Figure 16 compares the results for both the FEA and ParaPower analysis. The substrate based model correctly predicts the trend in stress as a function of h and has the same level of error, 12%, as the single run shown in figure 11. Run time for ParaPower was 1.1 seconds as compared to 485 seconds for the FEA.

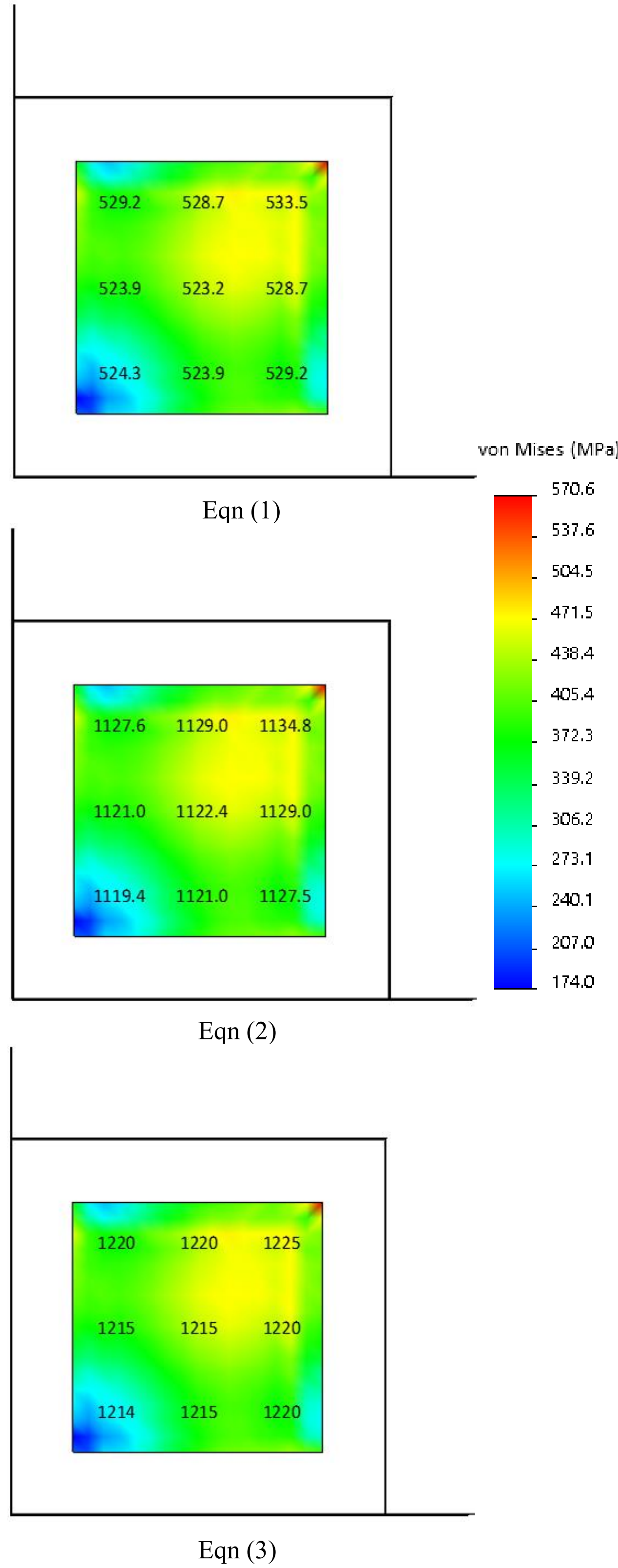


FIGURE 11: DEVICE COMPARISON

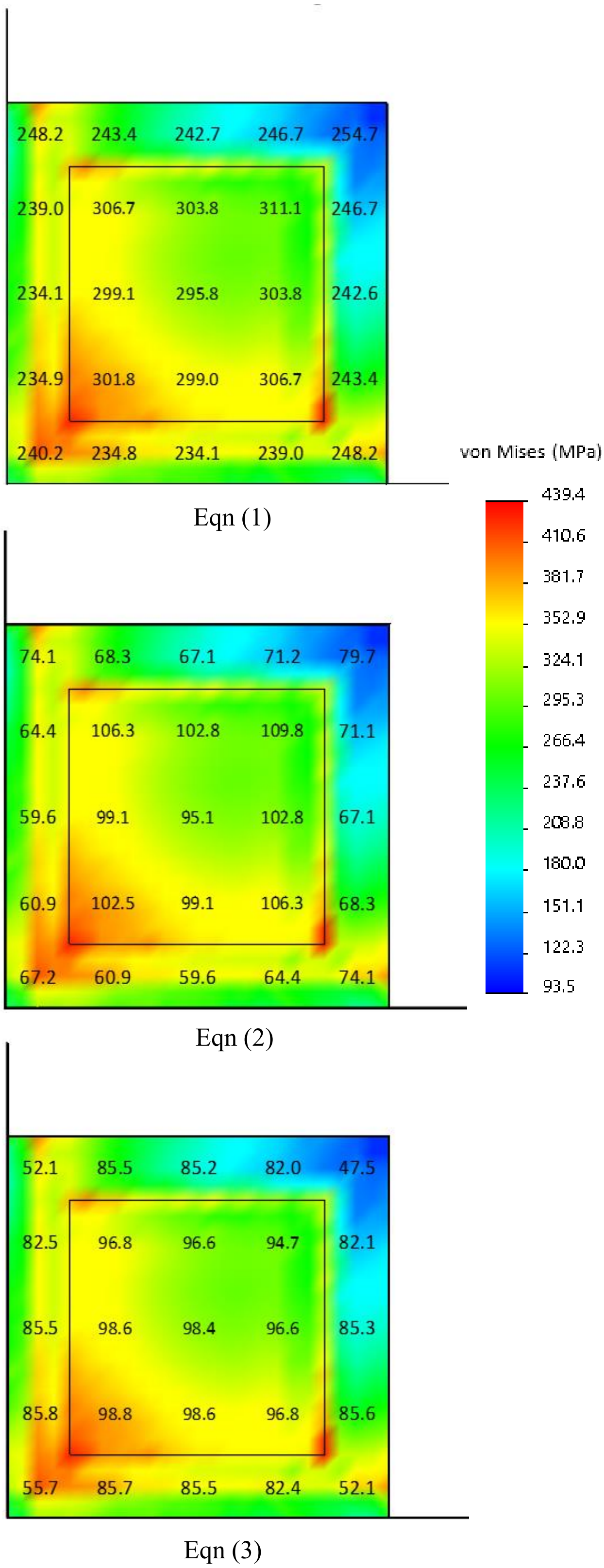


FIGURE 12: UPPER COPPER COMPARISON

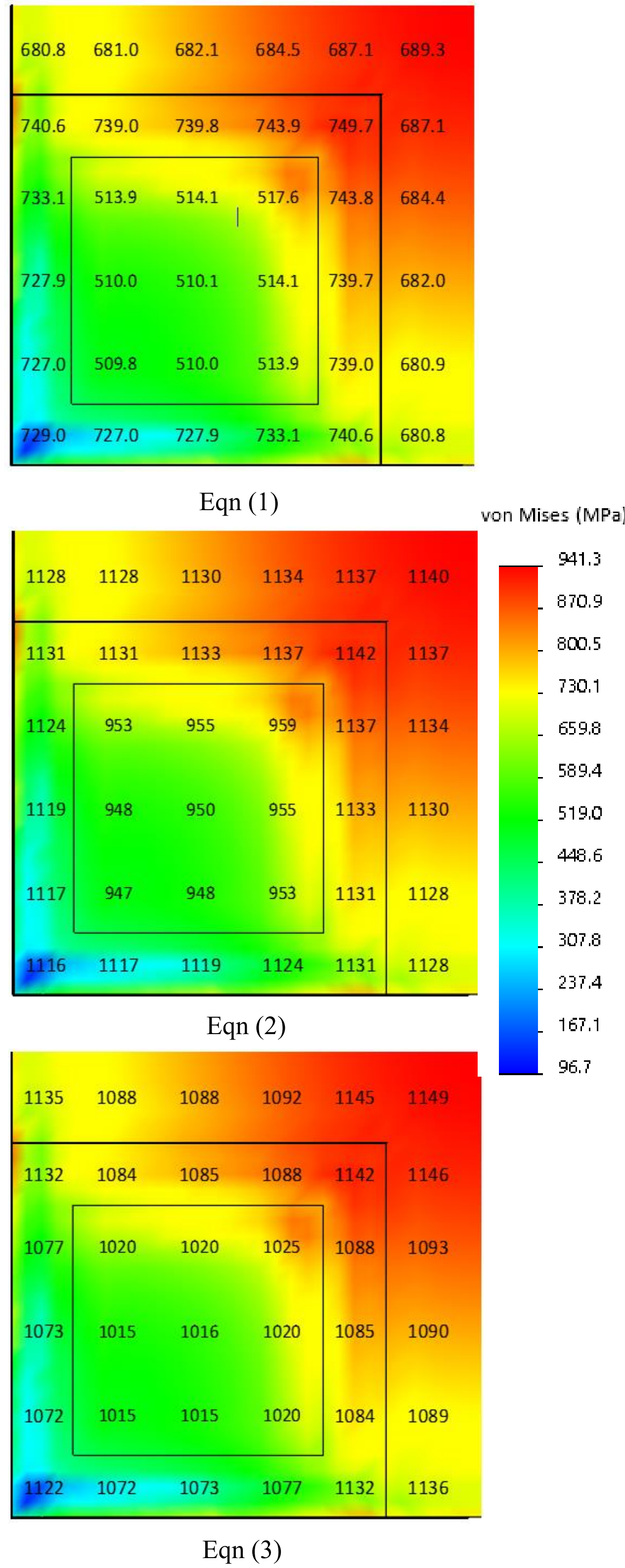


FIGURE 13: ALUMINUM NITRIDE COMPARISON

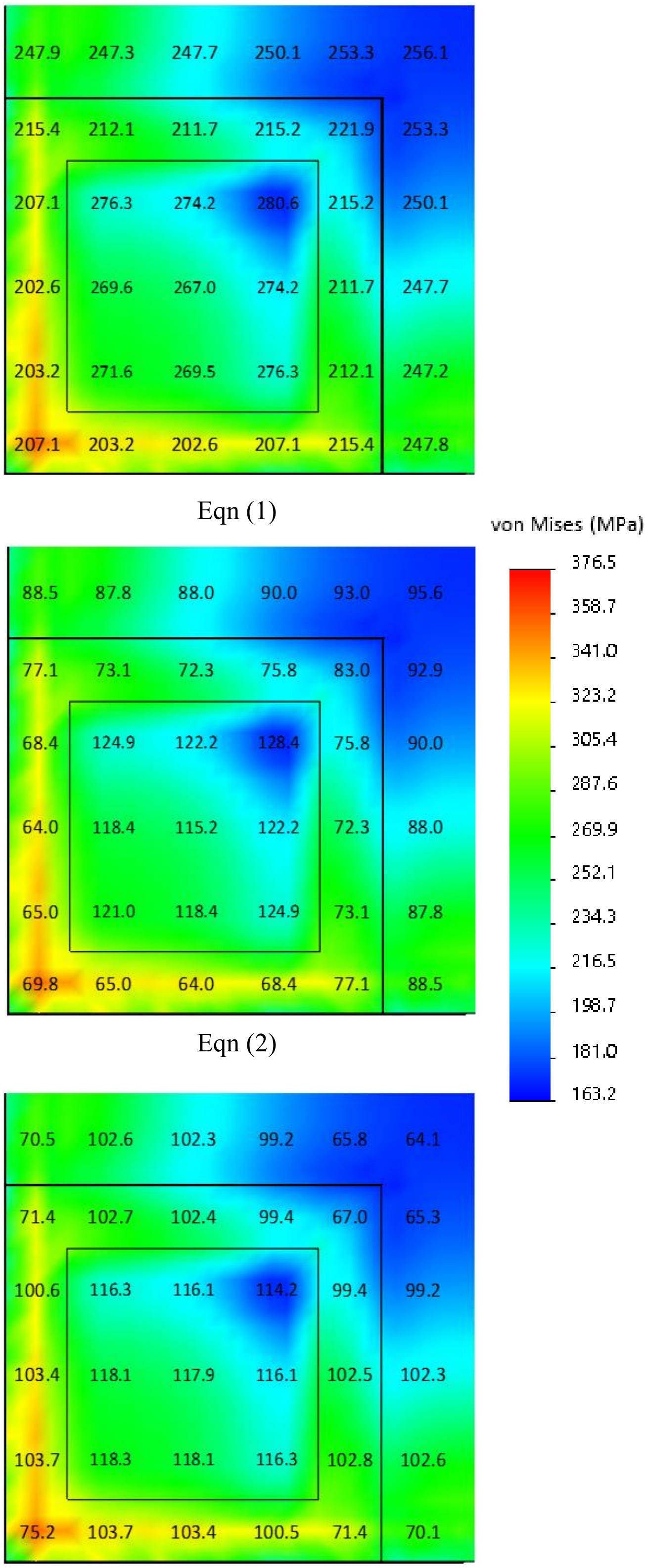


FIGURE 14: LOWER COPPER COMPARISON

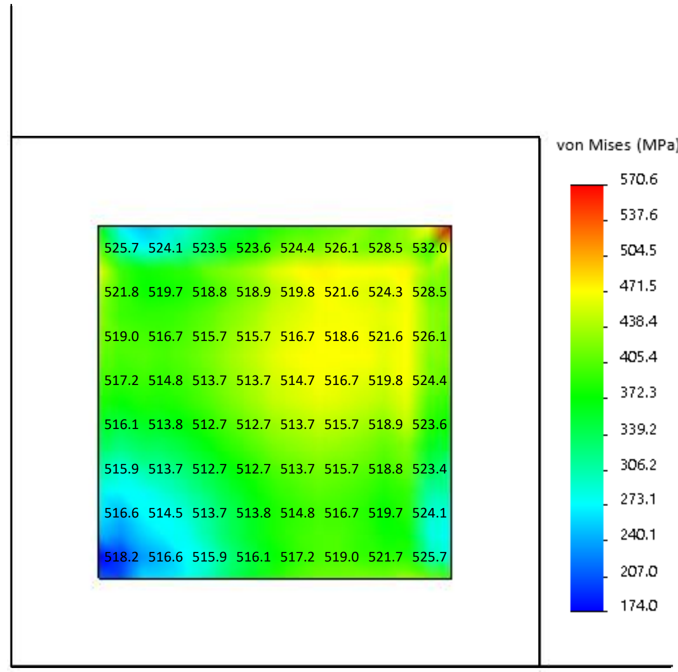


FIGURE 15: FINE GRID RESULTS, DEVICE LAYER

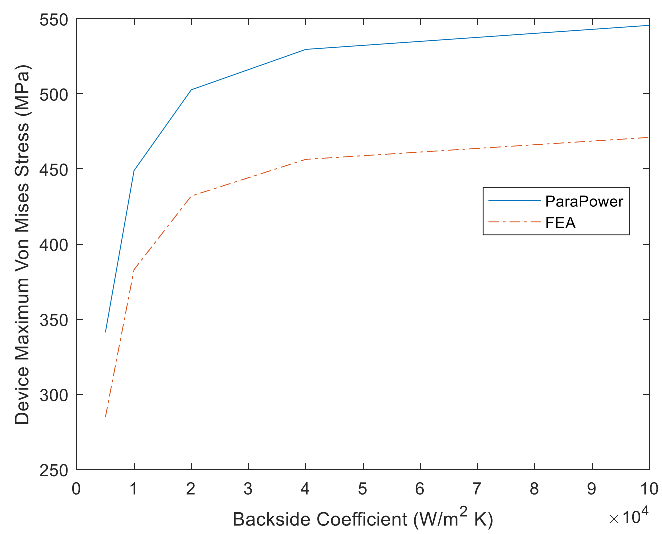


FIGURE 16: PLANAR MODULE PARAMETRIC RESULTS COMPARISON

4.2 Stacked Geometry

As with the planar geometry, results for the stacked geometry are presented for all three stress models. This geometry has two symmetry planes. Therefore, the results are displayed on only a quarter of the geometry. Figure 17 shows this region. The same labels are used here that were used in the planar geometry section. Comparisons are shown for the upper device, figure 18, and the center fin, figure 19. The magnitudes and trends for these locations are representative of the entire geometry. As with the planar module, comparisons are for areas just away from the peak corner stresses.

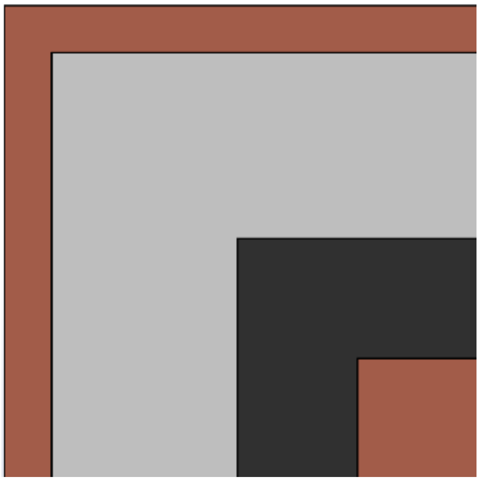


FIGURE 17: RESULTS DISPLAY REGION

Figure 18 shows that the substrate based and the first non-substrate based models greatly over predict the magnitude of the stress in the active part of the device. This is the area where the copper elements contact the device. Peak values from the FEA are approximately 350 MPA, but these two models predict values of roughly 1450 MPa, three times as large. The quasi 3-D model predicts values of just under 200 MPa for the active area of the device. In addition, the quasi 3-D model does a better job of predicting the stress distribution outside the active area of the device, with higher stress at the top and side of the active area and dropping off towards the upper left corner. The other two models show the opposite trend. Errors for the Eqn (1) and Eqn (2) models are roughly 250% for the peak value and 340% just away from the peak. For the Eqn (3) model, the corresponding errors are 51% and 35%. For the center fin, figure 19, the same trends hold. The quasi 3-D model does a better job of capturing the distribution of stress near the copper elements, higher stress near the edges. It also does a much better job of capturing the higher level of stress in the copper band that surrounds the assembly. The errors at this location for the Eqn (1) and (2) models are roughly 80% at the copper core and 100% at the copper band. The Eqn (3) model has errors of 10% and 18% at these same locations. Therefore, for geometries that have no clear substrate layer, the quasi 3-D model does a better job of capturing both the magnitude and distribution of stress.

Further evaluation of the quasi 3-D model included running the stacked geometry with a finer grid, 19 X 19 X 27, for 9,747 nodes. Again, the purpose of this run was to evaluate grid independence. Figure 20 shows the results for the device layer. Comparison to figure 18 shows that the magnitude and distribution of the stress predicted by ParaPower did not significantly change after roughly doubling the grid size.

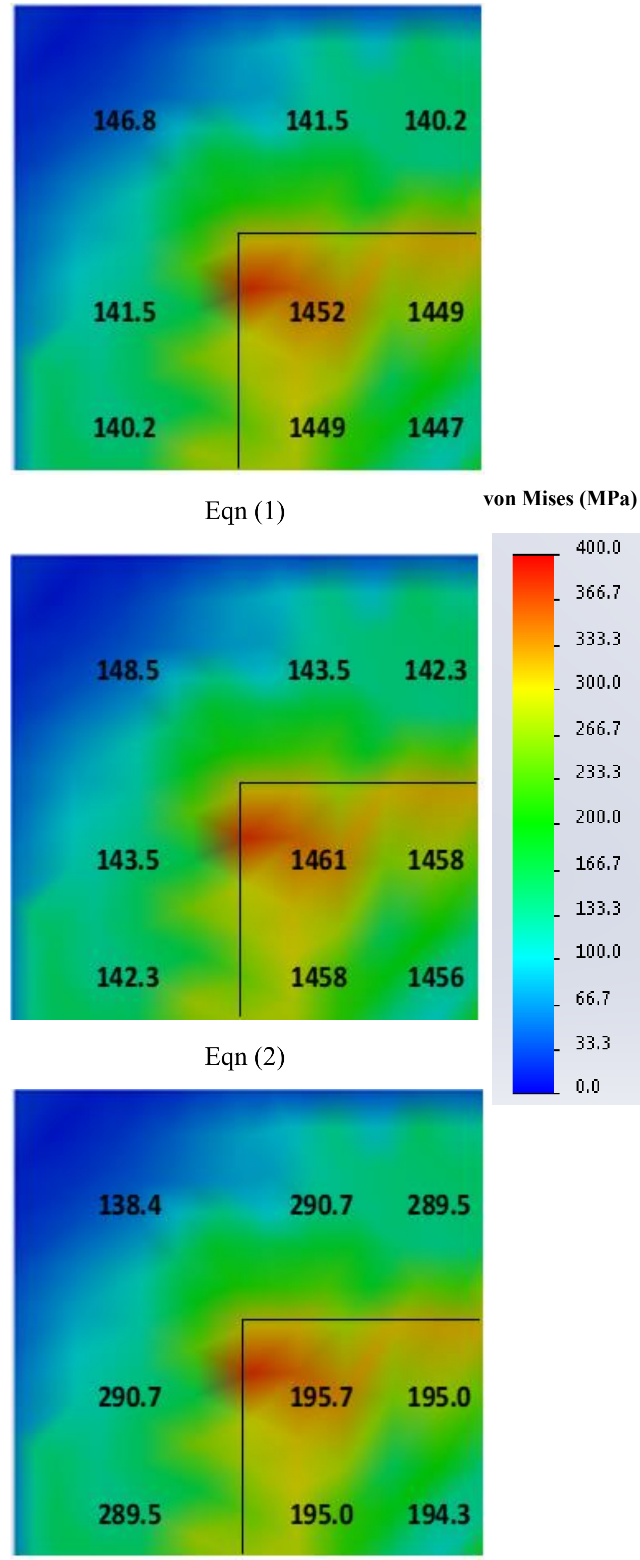


FIGURE 18: UPPER DEVICE COMPARISON

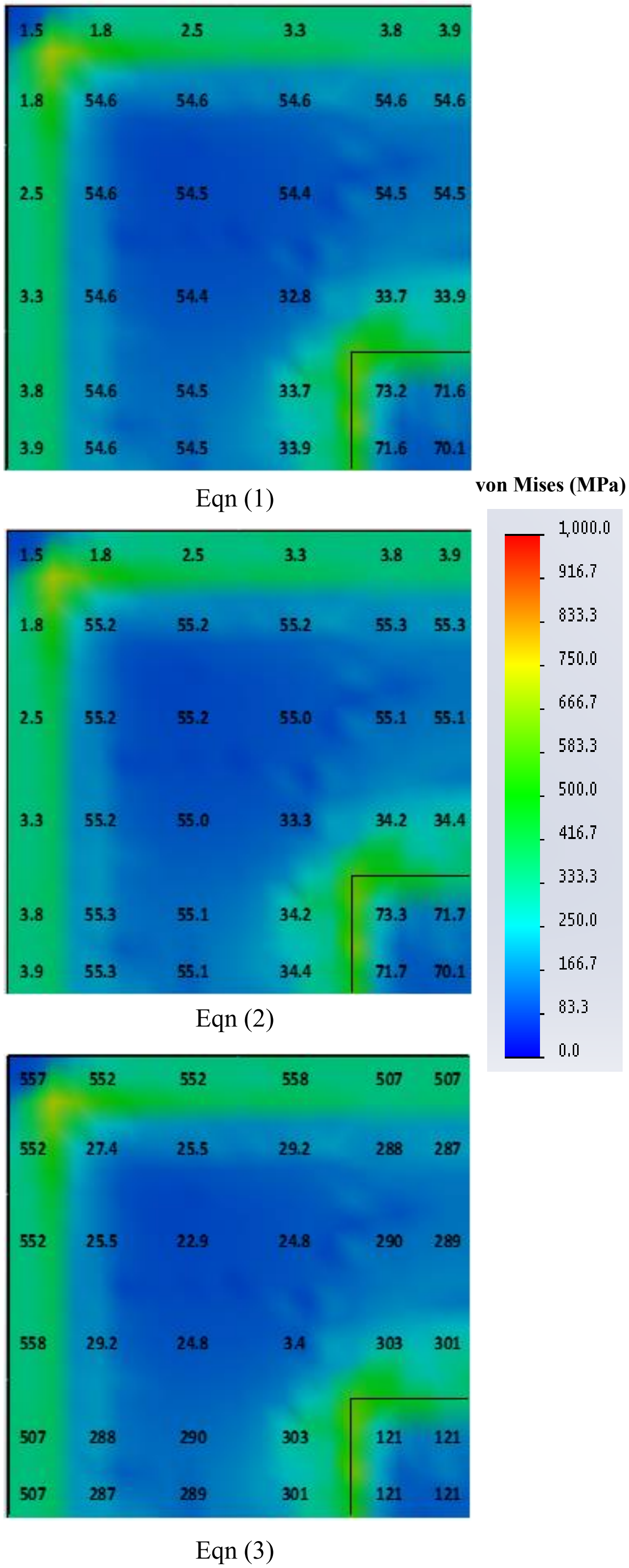


FIGURE 19: CENTER FIN COMPARISON

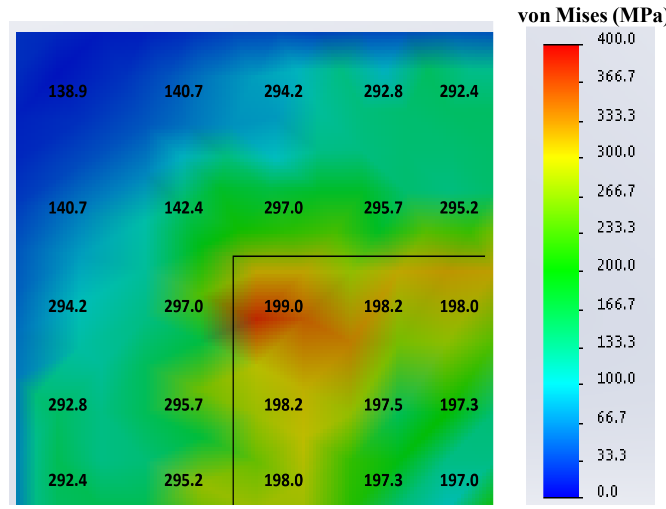


FIGURE 20: FINE GRID RESULTS, DEVICE LAYER

The final evaluation was a parametric analysis of the stress as a function of backside convection coefficient. Seven values of h were used, 1000, 2500, 5000, 7500, 10000, 12500 and 15000 W/m^{2}K. Figure 21 compares the results for both the FEA and ParaPower analysis. The quasi 3-D model correctly predicts the trend in stress as a function of h and has the same level of error, 51% and 31%, as the single run shown in figure 18. Run time for ParaPower was 2.3 seconds as compared to 405 seconds for the FEA.

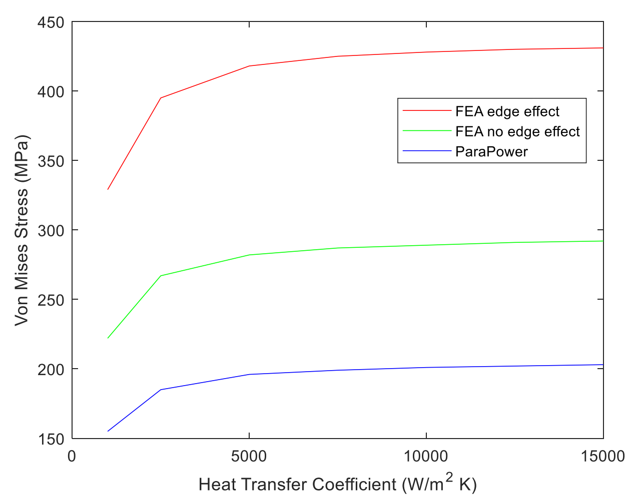


FIGURE 21: STACKED GEOMETRY PARAMETRIC RESULTS COMPARISON

5. CONCLUSIONS

The evaluation of both substrate based and non-substrate based models for thermal stress shows a need for both types of models. In the case where the geometry had a thick substrate layer, a typical planar module, the substrate based stress model was superior in predicting the magnitude of the stress. However, in the case of the stacked geometry, where there was no clear substrate layer, the quasi 3-D thermal stress model was superior in predicting the magnitude and distribution of stress. Errors for the substrate based model were less than 20% for the planar geometry and errors for the quasi 3-D model were roughly 30% for the stacked geometry. Both the substrate based and the quasi 3-D models are low order with short execution times, two orders of magnitude faster than the FEA for the same geometry, which makes them suitable for use in ParaPower. The goal in using ParaPower is to be able to adequately predict the temperatures and induced thermal stresses within the design space quickly. Once the design space has been evaluated, the most promising regions can be evaluated in detail.

ACKNOWLEDGMENTS

The authors would like to thank colleagues at the U.S. Army Research Laboratory who have helped support this effort including Bruce Geil and Morris Berman.

REFERENCES

[1] Boteler, L., Smith, A. "3D thermal resistance network method for the design of highly integrated packages", Proceedings of the ASME Summer Heat Transfer Conference, July 2013.

[2] Boteler, L. M., Miner, S. M., “Power Packaging Thermal and Stress Model for Quick Parametric Analyses,” Proceedings of the ASME 2017 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems, August 2017.

[3] Boteler, L. M., Miner, S. M., “Comparison of Thermal and Stress Analysis Results for a High Voltage Module Using FEA and a Quick Parametric Analysis Tool,” Proceedings of the ASME 2018 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems, August 2018.

[4] Boteler, L. M., Miner, S. M., Hinojosa, M., “Co-Designed High Voltage Module,” Proceedings of the IEEE 2018 ITHERM Conference, May 2018.

[5] Boteler, L. M., Miner, S. M., Fish, M., Berman, M., “Integrating Heat Sinks into a 3D Co-Design Network Model for Quick Parametric Analysis,” Proceedings of the IEEE 2019 ITHERM Conference, May 2019.

[6] Hsueh, C. H., “Thermal Stresses in Elastic Multilayer Systems,” Thin Solid Films, Vol. 418, Elsevier.