METHODOLOGY FOR THERMALLY INDUCED LOADING IN STRUCTURAL ANALYSIS ON THE F-35 LIGHTNING II

Byron E. Mueller[*] **and Robert M. Taylor**
Lockheed Martin Aeronautics Company

Keywords: *F-35 Lightning II, Thermal Loads, Finite Element Method*

Abstract

A program consistent methodology has enabled F-35 structural development teams to effectively and consistently develop and include thermally induced loads in structural analyses. A study correlating finite element model results to test data formed the basis for the thermally induced loads methodology. This study calibrated a global finite element model, which used sprung skins and temperature dependent material properties, with large scale component test data from hot and cold soak test conditions. The correlation effort established empirical calibration factors and demonstrated the feasibility of a global air vehicle approach to thermally induced load development. Global internal thermal loads were developed using the air vehicle finite element model based on these results.

1 Introduction

Structural sizing and analysis on a high performance fighter program must consider thermally induced stresses and deflections due to aircraft heating and cooling effects individually and together with other airframe loading conditions, depending on which is more critical. All failure modes normally considered for mechanical only loading must be considered for ground thermal soak conditions and mechanical-plus-thermal loading conditions. The Department of Defense Joint Service Specification Guide (JSSG-2006) [1] provides guidance on aircraft structures design criteria to be used in the development of performance specifications. This guide indicates that airframe limit loads must address, among other considerations, "heat flux and the thermal strains resulting from the resulting temperature gradients." Additionally, the guide specifies that "where thermal loads are significant, factors of uncertainty to apply to the external or internal thermal loads should be specified."

Accordingly, the F-35 Lightning II Development Program defined a methodology to account for thermal loads both locally and globally. Local thermal loading effects, usually due to aircraft systems in the aircraft that either generate heat or provide cooling to the structure, can be addressed by applying temperature gradients to the component or region of interest through classical analysis or local fine grid finite element (FE) analysis methods. Global structural response is not required in these local analyses and the methodology is well defined historically. On the contrary, global thermal loading effects due to coefficient of thermal expansion mismatch requires determination of global structural response under thermal conditions. Implementation of the global methodology requires correlation of global finite element model results to large-scale test data and calibration of the global finite element model so it produces structural response within an acceptable range of the test data.

This paper describes the program consistent methodology used to address global thermally induced loading in static structural analysis on the F-35 Lightning II Development Program. The paper first presents background material on thermally induced loading relevant to the current discussion. Next, it outlines a correlation study used as the basis for the F-35

[*] Formerly with Lockheed Martin Aeronautics Company

methodology. It then describes the thermal loading air vehicle finite element model and conditions used for thermally induced loading. Finally, the paper provides an overview of the thermal loading stress analysis procedure. While methodology was also developed for addressing thermally induced loads in F-35 durability and damage tolerance (DaDT) analysis, the scope of this paper is limited to discussion of the static analysis methodology only.

2 Background

Thermally induced loads arise due to global heating and cooling of the airframe structure caused by ambient air and aircraft maneuvering effects, or due to local heating and cooling effects caused by the operation of systems within the aircraft. Cases where thermally induced stresses are most significant are:

1) When coefficients of thermal expansion (CTE) between parts fastened and/or bonded together are significantly different such that temperatures above or below the fastened assembly temperature (room temperature), or bonded assembly adhesive gel temperature, result in a different expansion or contraction of the two parts. This results in relative loading between the parts. An example would be graphite composite skins mechanically fastened to aluminum substructure.

2) When structure subjected to local heating or cooling effects is attached between structure that is relatively rigid such that expansion or contraction of the part is restrained, resulting in significant thermally induced loads.

3) When structure subjected to local heating or cooling over a portion of the structure is such that thermal gradients develop across the part resulting in locally induced thermal loads. These loads are caused by the restraining effects of the surrounding material within the part itself.

As indicated previously, local thermal effects (cases 2 and 3 above) can be treated using locally applied temperature gradients in a local analysis and will not be addressed in the current discussion of global methodology.

Classical theory of elasticity [2] and structures [3] texts provide supporting theory for including thermal expansion based on linear coefficients of expansion and delta temperatures ($\alpha\Delta T$) in linear combination with mechanical loads. This theory forms the foundation for thermal loads development through computational finite element methods. [4]

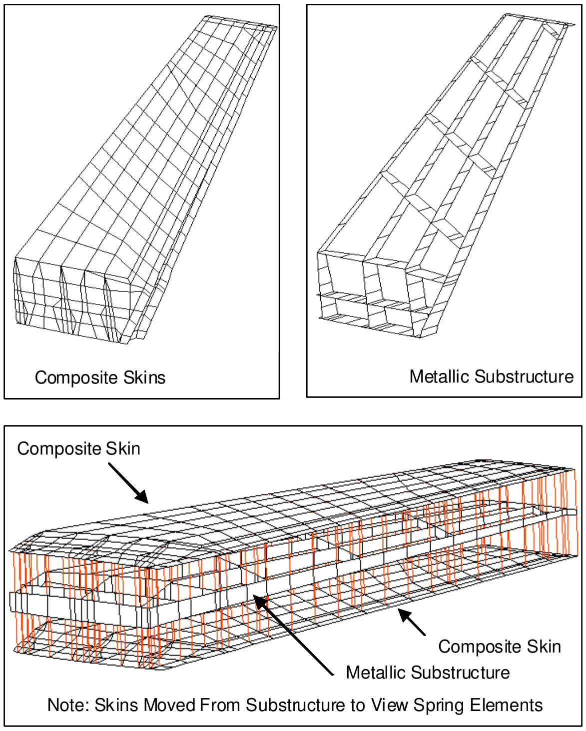
A significant body of work exists in the area of thermal structures. Thornton [5] surveyed work and historical developments in thermal structures from the 1950s to the 1980s. His survey included aerodynamic heating, aerothermal load effects on flight structures, design of thermal structures, heat transfer and thermal stresses. While the current discussion addresses the global effect of CTE mismatch, work in the literature related to CTE mismatch has focused only on local effects, such as the CTE mismatch between composite fiber and matrix material [6], composites with metallic coatings [7] [8], and bonded repairs [9].

Yang, et al [10], developed an analytical model to address thermally induced loads due to CTE mismatch in aluminum and composite hybrid structure and correlated the model to test. This study notably included effects due to the length of aluminum beams on the thermally induced stresses. The study looked at a through thickness temperature gradient representative of a differential between outside air temperature and an insulated internal cabin temperature on a transport aircraft. In contrast, military fighter configurations would not have such a through thickness gradient but would rather use thermal soak conditions given the absence of insulating materials.

3 Correlation Study

Given structural design requirements and the aircraft operating environment, accurate representation of the thermally induced loads is critical to ensure both structural integrity as well as minimum weight. A study correlating finite element model results to test data formed the basis for the thermally induced loads methodology. The purpose of this study was to determine the viability of using a finite element model approach to analyze assemblies of composite skin and metallic substructure subjected to temperature loading. The study examined a built-up box configuration of an actual aircraft assembly with graphite epoxy skins mechanically fastened to aluminum alloy substructure. Strain gage data already existed from tests that subjected this assembly to various hot and cold soak temperature conditions. Strains due to thermal mismatch that resulted in thermally induced stresses were derived from the strain gages that had been bonded at the same locations and in the same directions on the mating skin and substructure parts. Free expansion strains that do not contribute to thermally induced loads were filtered from the test strains using bridge circuits that included gages on coupons manufactured using the same material but were allowed to free expand or contract under the applied temperature conditions. Similarly, strains obtained directly from finite element results were adjusted to eliminate free expansion strains that did not contribute to thermally induced loads by subtracting calculated free expansion strain values from the resulting strain results.

Earlier finite element correlation studies had indicated that the coarse grid finite element model results predicted thermal loading effects that were too conservative, i.e. the magnitude of predicted strains was significantly higher than the measured strains. In the correlation study, two modeling adjustments were made to reduce the observed conservatism in the finite element thermal modeling approach. The first adjustment addressed fastener joint modeling. The fastener joints in the coarse grid finite element models were through rigid grid point attachments that did not properly account for joint flexibility. The correlation study investigated the improvement in finite element model results correlation that could be achieved by adding springs to better simulate the actual joint flexibilities. A finite element model of the test configuration, shown in Figure 1, used shell elements with composite skins and metallic substructure.



Note: Skins Moved From Substructure to View Spring Elements

Figure 1: Correlation Study Finite Element Model

Composite skins in the model were sprung in shear to account for flexibilities between skin and substructure. Spring stiffness values were calculated using a Lockheed Martin developed tool based on fastener type, fastener size, fastener spacing, number of fastener rows, mating structure thicknesses, and mating structure moduli. Since coarse grid models will not have grid points that line up with fastener locations, spring stiffnesses are smeared values that account for the difference in fastener spacing and the number of fastener rows compared to the grid point spacing and the single row of grids that usually exist at coarse grid model skin-to-substructure joints. MSC/NASTRAN CBUSH elements are used to model the fastener joint where stiffness values are provided in all three translational degrees of freedom and two rotational degrees of freedom. The CBUSH elements use a local coordinate system that is oriented with the actual structure, as shown in Figure 2, and are oriented in the fastener row direction (coordinate 1), normal to the fastener row direction in the plane of the skin (coordinate 2), and normal to the plane of the skin (coordinate 3). If the skin curves at the spring location then the local skin tangential planes are used to determine the plane of the skin for orienting the CBUSH. The finite element representation of the spring joint is shown in Figure 3.

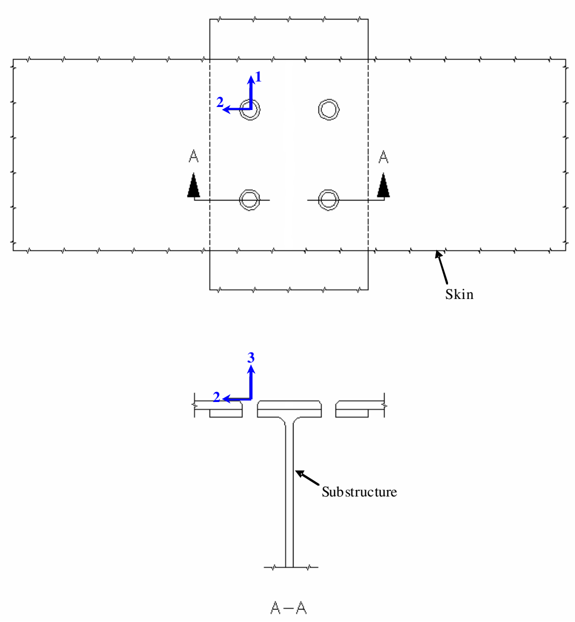


Figure 2: CBUSH Spring Definition and Orientation

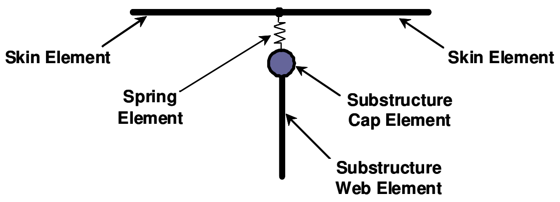


Figure 3: Finite Element Representation of Sprung Skin

The primary concern for thermal loading is in the substructure flange direction (coordinate 1) since this is the direction where relative expansion effects between the skin and substructure result in the largest build up of thermal loads. Spring stiffness values in the coordinate 2 direction are not as critical, and therefore are set equal to the coordinate 1 calculated spring stiffness in order to reduce the finite element modeling task.

The two rotational degrees of freedom correspond to the two CBUSH orthogonal directions that are in-plane with the skin, coordinates 1 and 2. These rotational degrees of freedom are set to essentially rigid stiffness values to provide model stability and a more realistic joint configuration. The rotational degree of freedom stiffness in the direction normal to the skin, coordinate 3 or fastener axial direction, is set to zero.

The second modeling adjustment addressed material properties. The dependency of material physical properties on temperature is well documented in industry standard specifications such as MIL-HDBK-5H [11]. As an example, Table 1 gives CTE values for 2024 aluminum alloy at various temperatures as given in MIL-HDBK-5H. The previous studies assumed constant values for CTE and these assumptions proved to be contributing factors to the conservative results obtained. In this study, CTE values used in the finite element model were adjusted to actual values at the modeled temperatures.

Table 1: Effect of Temperature on Coefficient of Thermal Expansion of 2024 Aluminum Alloy (source MIL-HDBK-5H)

Temperature, °F	Coefficient of Thermal Expansion (α), 10^{-6} in./in./°F
-65	11.8
0	12.2
75	12.6
220	12.9
350	13.3

The finite element model study was conducted in three stages to determine the impact of each adjustment. Initially a coarse grid finite element model was analyzed without the addition of springs between the skins and substructure, and correlated to the test results. Then the model was revised to include spring elements between the skin and substructure parts and correlated with the test results. Finally, temperature adjusted CTE values were incorporated in the model and the results were again correlated to the test results.

The model with sprung skins exhibited good correlation to the test data as evidenced by the comparison of finite element results to test data as shown in Figures 4 through 7 below. Accounting for the variation of thermal expansion coefficient further improved the results as shown for the cold soak condition on the aluminum structure gages.

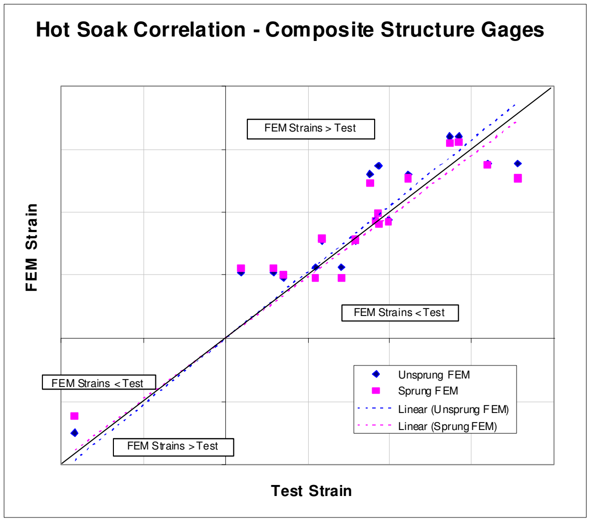


Figure 4: Correlation Study Finite Element Results vs. Test Data—Hot Soak Composite Structure Gages

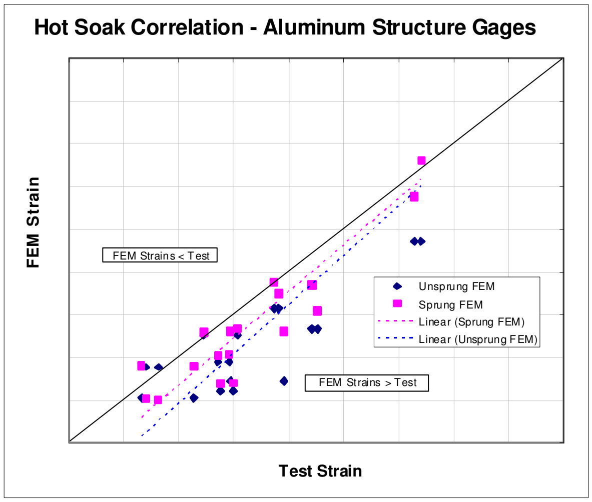


Figure 5: Correlation Study Finite Element Results vs. Test Data—Hot Soak Aluminum Structure Gages

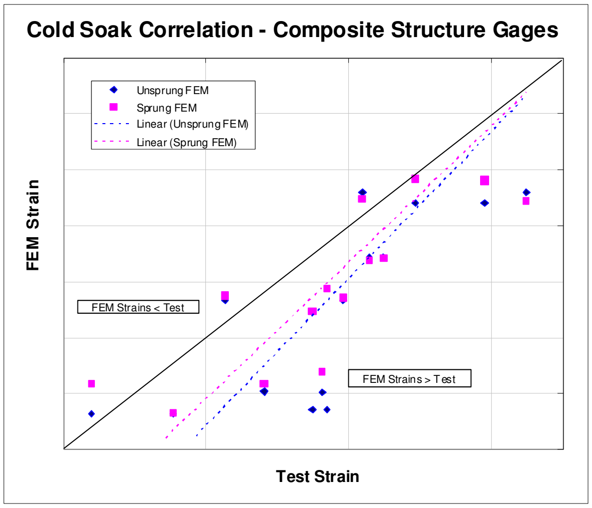


Figure 6: Correlation Study Finite Element Results vs. Test Data—Cold Soak Composite Structure Gages

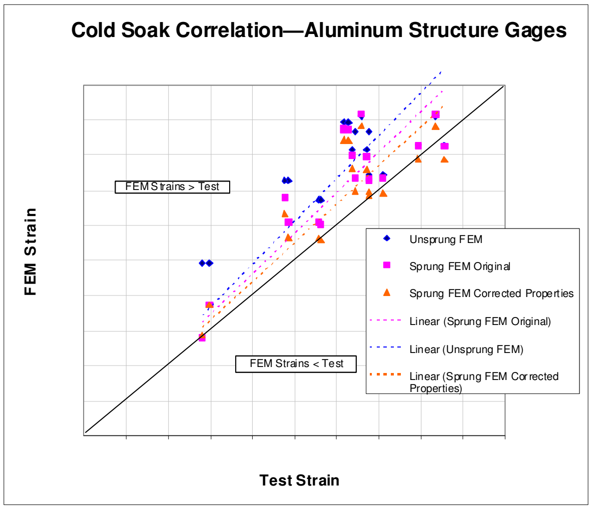


Figure 7: Correlation Study Finite Element Results vs. Test Data—Cold Soak Aluminum Structure Gages

In addition to the improved correlation due to the modeling adjustments, the correlation effort indicated a sensitivity of thermal load to part length, showing that thermal loading is reduced as the length of the substructure part is reduced. This study only considered the effect of modeling joints between the skins and substructure parts; however, a similar effect occurs due to the fastener joints between the substructure parts since the finite element model effectively analyzes these parts as long continuous parts, increasing the thermal loading effect. Since it is not practical to add springs to fastener joints between substructure parts when modeling coarse grid models, a part length correction factor was developed for reducing conservative predictions for short parts that are joined together so as to create long parts in the finite element model.

Another adjustment made in the F-35 thermal loads methodology was to model skin splices as actual breaks in the skin elements at the splice locations; however, the test article used for correlation did not contain skin splices, and therefore this part of the proposed analysis approach was not included in this study. With skin splices, the CBUSH definition remains the same, as shown in Figure 8, and the finite element representation of the skin-to-substructure joints require two springs, as shown in Figure 9.

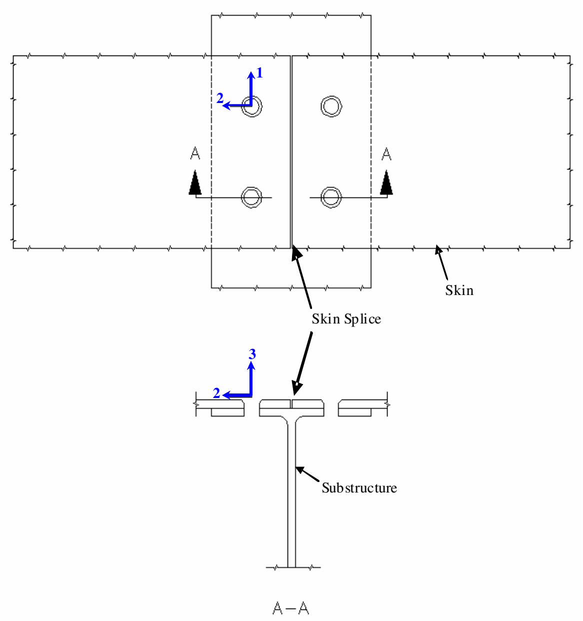


Figure 8: CBUSH Spring Definition and Orientation

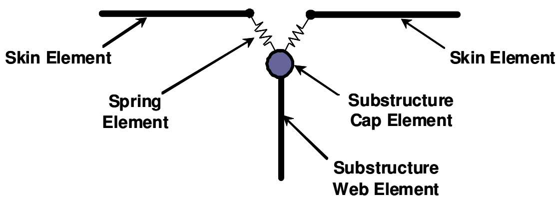


Figure 9: Finite Element Representation of Sprung Skin with Skin Splice

This correlation study established finite element model configuration, determined empirical calibration factors, and demonstrated the feasibility of a global air vehicle approach to thermally induced load development. These calibration factors were applied to development of the F-35 air vehicle thermal loading finite element model as described in the next section.

4 Thermal Loading Finite Element Model

The second aspect of the program consistent thermal loads methodology is development of airframe internal loads due to thermal loading of global heating or cooling of the airframe structure. These loads are determined using the appropriate F-35 air vehicle coarse grid finite element model results, shown in Figure 10, for combined mechanical-plus-thermal loading analysis.

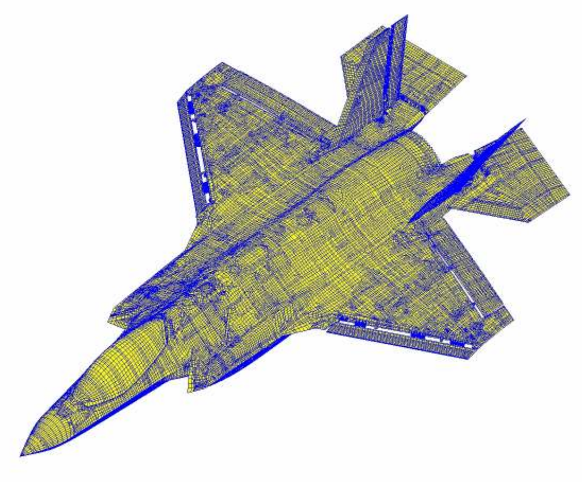


Figure 10: Air Vehicle Finite Element Model Used for Thermal Loading

The thermal FEM is separate but nearly identical to the mechanically loaded FEM. Based on the results of the correlation study described previously, some modifications have been made to account for the flexibilities within the structure that have a significant reduction effect on thermally induced internal loads. Additionally, material properties have been modified to account for changes in the thermal expansion coefficient properties as a function of temperature, further reducing conservatism in the thermal loading analysis results. Element IDs for airframe structure are consistent between models to enable the combination of thermally induced and mechanically induced element internal loads. This allows the structural analyst to combine thermal and mechanical load datasets for determining the combined loads. Combining grid point force balance data from the thermal loads and mechanical loads models could not be performed directly at the locations where additional CBUSH spring elements and the additional grid points were included in the thermal loads model to simulate the fastener joints and skin splices. Internal loads processing programs at Lockheed-Martin were modified to process data at these locations so that the grid point force data could be combined as if the finite element models were both modeled without the springs and skin splices. Strain data was not recovered from the thermal loads model runs since no adjustment was made to account for free expansion strains, and therefore, strain data recovered from the thermal loads model would not have accurately represented the internal loads induced by thermal loading.

Four basic types of thermal load cases are applied to the thermally loaded finite element models: 1) hot soak ground condition, 2) cold soak ground condition, 3) hot flight maneuver condition and 4) cold flight maneuver condition. The correct thermal load condition and mechanical load condition combination is made when performing critical load case searches using a database tool and obtaining internal loads data for analysis. Thermal induced loads that relieve mechanical loads are neglected.

In practical application of the thermal loading methodology, the engineer first obtains part geometry and FEM information, as shown in Figure 11 for a flange on a structural component. Next, temperature exposure for the part is determined based on applicable thermal load cases, as shown in Figure 12. If the part temperature exposure is within a range where thermal loading is known to be insignificant, as established by program policy, thermal loading can be neglected and critical load cases are determined using mechanical only loads. If the part temperature is outside this range, thermal loading must be considered and the next step is to determine if a part length correction factor applies based on gross part length, as shown in Figure 13. If so, the value of the correction factor is determined based on a curve, shown in Figure 14, that was developed from the previously discussed correlation study. Critical load cases are then determined using mechanical plus thermal loads, applying the part length correction factor to thermal conditions if appropriate.

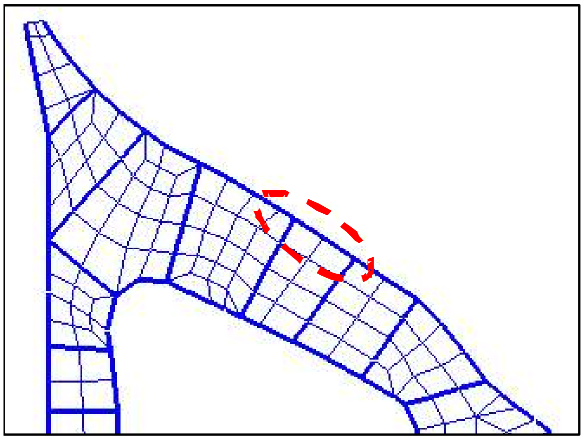


Figure 11: Part Finite Elements Used in Determining Critical Thermally Induced Internal Loads

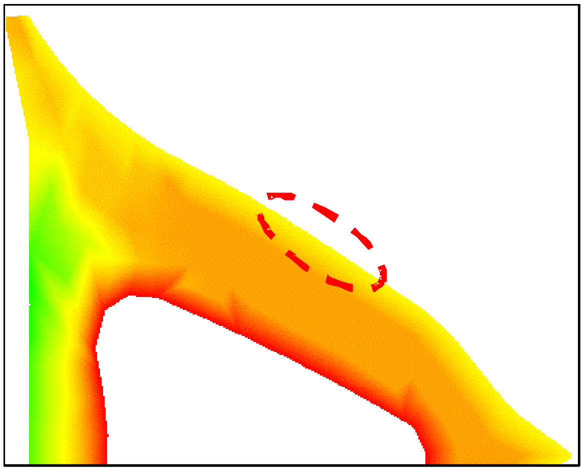


Figure 12: Temperature Used in Determining Critical Thermally Induced Internal Loads

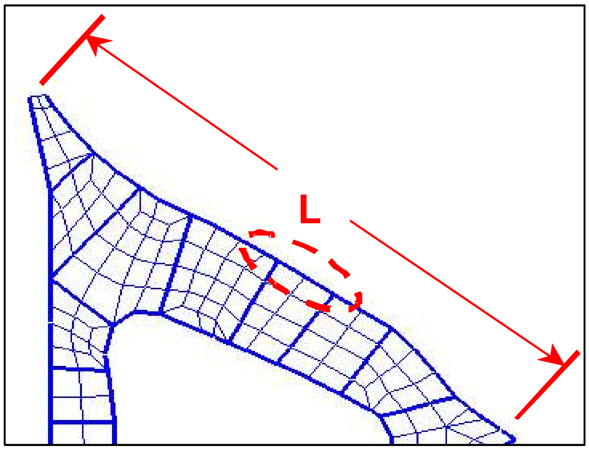


Figure 13: Part Length Used in Determining Critical Thermally Induced Internal Loads

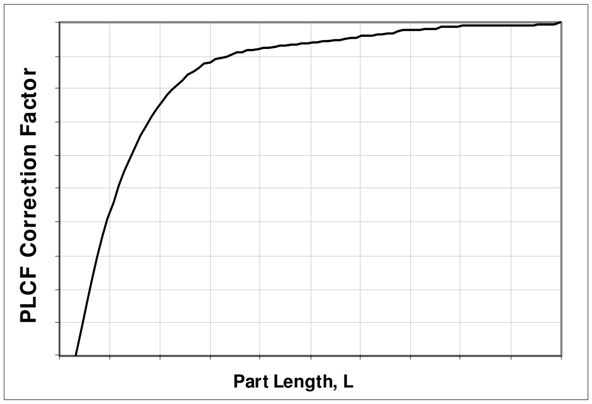


Figure 14: Part Length Correction Factor Used in Determining Critical Thermally Induced Internal Loads

3 Conclusion

The methodology described herein has enabled F-35 structural development teams to effectively and consistently include thermally induced loads in structural analyses to satisfy structural design requirements. The methodology was validated through a correlation study that calibrated finite element modeling with test data. Adjustments to the finite element model, which included spring element attachments of skin to substructure and temperature adjustment of the coefficient of thermal expansion, played a key role in correlating finite element results to test data. With these adjustments, the model was calibrated to test data and a methodology was developed for using the global air vehicle finite element model for incorporating thermally induced loads in detail part structural analysis. The global air vehicle model approach allows critical load case determination to consider the effects of just mechanical loading or mechanical plus thermal loading, thereby ensuring the structural design requirements are met.

References

[1] JSSG-2006, *Department of Defense Joint Service Specification Guide, Aircraft Structures*, 1998.

[2] Timoshenko, S.P., and Goodier, J.N., *Theory of Elasticity*, McGraw-Hill, 1951.

[3] Gatewood, B.E., *Thermal Stresses*, McGraw-Hill, 1957.

[4] Cook R, Malkus D, and Plesha M., *Concepts and Applications of Finite Element Analysis*, Wiley, New York, 1989.

[5] Thornton, E. A., “Thermal Structures: Four Decades of Progress,” *Journal of Aircraft*, Vol. 29, No. 3, May-June 1992.

[6] Naik, R. A., Crews, J. H., *Closed-form Analysis of Fiber-matrix Interface Stresses under Thermo-Mechanical Loadings*, NASA-TM-107575, 1992.

[7] Rousseau, C. Q., Hyer, M. W., Tompkins, S. S., *Stresses and Deformations in Angle-ply Composite Tubes*, NASA-TM-89365, 1987.

[8] Knott, T. W., Hyer, M. W., *Analytical Study of Graphite-Epoxy Tube Response to Thermal Loads*, NASA_CR-183215, 1988

[9] Daverschot, D. R., Vlot, A., and Woerden, H. J. M., “Thermal Residual Stresses in Bonded Repairs,” *Applied Composite Materials*, Vol. 9, No. 3, 2002.

[10] Yang, C, Sun, W, Seneviratne, W, and Shashidhar, A, “Thermally Induced Loads of Fastened Hybrid Composite/Aluminum Structures,” *Journal of Aircraft*, Vol. 45, No. 2, March–April 2008.

[11] MIL-HDBK-5H, *Metallic Materials and Elements for Aerospace Vehicles and Structures*, 1998.

Copyright Statement