Using FloTHERM XT and ANSYS Workbench to Perform Thermo-Mechanical Analysis

Kenneth C. Nwanoro, Hua Lu, Chunyan Yin, Chris Bailey, Computational Mechanical & Reliability Group, University of Greenwich

Thermally induced stress in electronic products is a growing concern for electronic package designers. These stresses result in material degradation of the package, and lead to a wear-out mechanism known as fatigue. Depending on the magnitude of these stresses, fatigue can result in premature loss of performance and hence reliability of the overall product. This impact of fatigue is governed by (i) system design (ii) non-uniform temperature distribution during operation and (iii) mismatch in materials properties (e.g. co-efficient of thermal expansion (CTE)).

Accurately predicting the magnitude of stress requires a coupled thermo-mechanical analysis. First, the non-uniform temperature distribution across the system needs to be predicted, taking into account the relevant heat sources and heat transfer mechanisms. Secondly, the resulting deformation, strain (including creep) and stress distributions across the package needs to be predicted.

Best-in-class analysis tools used by electronic product designers treat thermal and mechanical (Stress) design separately. In particular, CFD tools are used for thermal analysis and FEA tools are used for mechanical design. Although thermo-mechanical analysis, using FEA tools, is not new, the majority of reported work assumes that temperature changes across the package/system are constant. Of course, in reality, this is not the case. Current trends in both micro and power electronics packaging include the use of 3D-ICs, Through Silicon Vias, Wafer-Level Packaging, Higher switching frequency, etc, and operating in higher temperature environments. Hence there is an increasing need to enable thermal and mechanical design engineers to work collaboratively using best-in-class analysis tools.

In this article we will discuss the workflow options for using Simcenter Flotherm XT thermal results to perform thermo-mechanical analysis using ANSYS® Workbench™. This is illustrated with the following two case studies:

- Case 1: BeagleBone™ – Flotherm XT is used to predict the steady state temperature distribution across the whole board. The temperature distribution for a particular component is then mapped into ANSYS to undertake thermo-mechanical analysis.
- Case 2: IGBT – Flotherm XT is used to predict transient temperature distribution due to power cycling. These temperatures across the insulated gate bipolar transistor (IGBT) power module are then mapped into ANSYS Workbench to perform thermo-mechanical analysis.

The FloTHERM XT is finite volume based computational fluid dynamics (CFD) software that is suitable for predicting temperatures in electronics systems. It has in its library many detailed electronic smart parts that enable easy thermal and flow simulation of electronic systems and components. ANSYS Workbench is a finite element analysis (FEA) software that is suitable for performing many multi-physics simulations such as thermo-mechanical analysis. The two software are used so that their advantages can be exploited to maximise the efficiency and accuracy.

Workflow

The workflow used to map FloTHERM XT results into ANSYS Workbench is quite straight forward and sequential. This is because both software tools have the capability to import and export external data and ANSYS Workbench has mapping functions with options to transfer external data across two dissimilar mesh interfaces.

- Step 1**: Run Flotherm XT**: First, a detailed thermal or/and fluid flow analysis is performed using Flotherm XT using a CAD geometry that is created in the software or imported.
- Step 2: **Export Flotherm XT Results**: Analysis files from FloTHERM XT (Temperature results) are exported at every time step specified (if transient analysis) using the **Export Results** function of FloTHERM XT in a text file format.
- Step 3: **Build Model in ANSYS**: The same model geometry is created in ANSYS Design Modeler or imported CAD geometry is meshed in ANSYS Workbench. It is necessary to use the same coordinate system and units in the two software. FloTHERM XT exports results in delimited format and semicolon delimiter

type as well as temperature results in Kelvin and these should be used in the ANSYS Workbench External Data settings.

- Step 4: **Import FloTHERM XT Results**: The exported FloTHERM XT results (step 2, above) are imported using the External Data system in ANSYS Workbench and mapped across to the FEA mesh. Appropriate boundary conditions are applied after which the model is solved and the stress results analysed.

Case One Predicting the temperature and stress/strain distribution in BeagleBone components under typical operating loading conditions.

The BeagleBone is an open source single board computer system that is used for exploring hardware and software capabilities, often for educational and hobbyist purposes. This case study explores the usefulness of combining two numerical software to obtain steady state system level temperature distribution and then using the temperature results to obtain board level stress and strain distribution.

A detailed thermo-mechanical analysis at component level, such as solder joints in a ball grid array (BGA) while considering material nonlinearities in order to predict the damage occurring during system operation is performed using a quarter model of the beagle bone board due its symmetry.

The Beagle bone model used is shown in the Figure 1. Figure 1a, b and c are respectively the enclosed beagle bone CAD geometry, the detailed board assembly and the heat sink –BGA assembly. The model includes a top and bottom plastic (Nylon 66) enclosure, an aluminium heatsink mounted on top of the detailed BGA and a series of chips (ICs) modelled as simple cuboids. The PCB board has been modelled in detail using the FloTHERM XT PCB Non-Geometric Smart Part in order to replicate the characteristics of the actual system structure. The detailed steady state thermal analysis is performed using FloTHERM XT. Typical power dissipation ranging between 0.1 and 0.6W is applied in each chip and 35°C ambient temperature is assumed. For the structural analysis, constraint to prevent free body movement is applied and the whole structure is assumed stress free at 35°C. The results are shown in Figure 2.

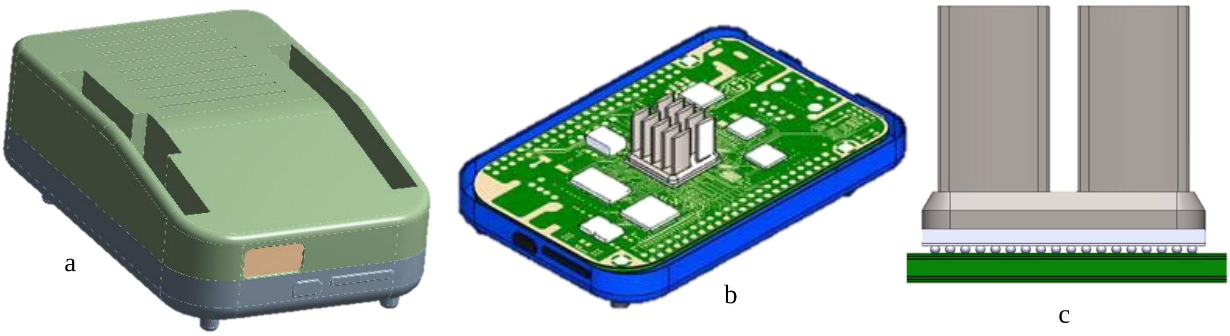

Figure 1. Beagle bone model

Figure 2 a and c show the FloTHERM XT system and board level temperature distributions respectively while 2 b and d show the imported temperatures in ANSYS Workbench.

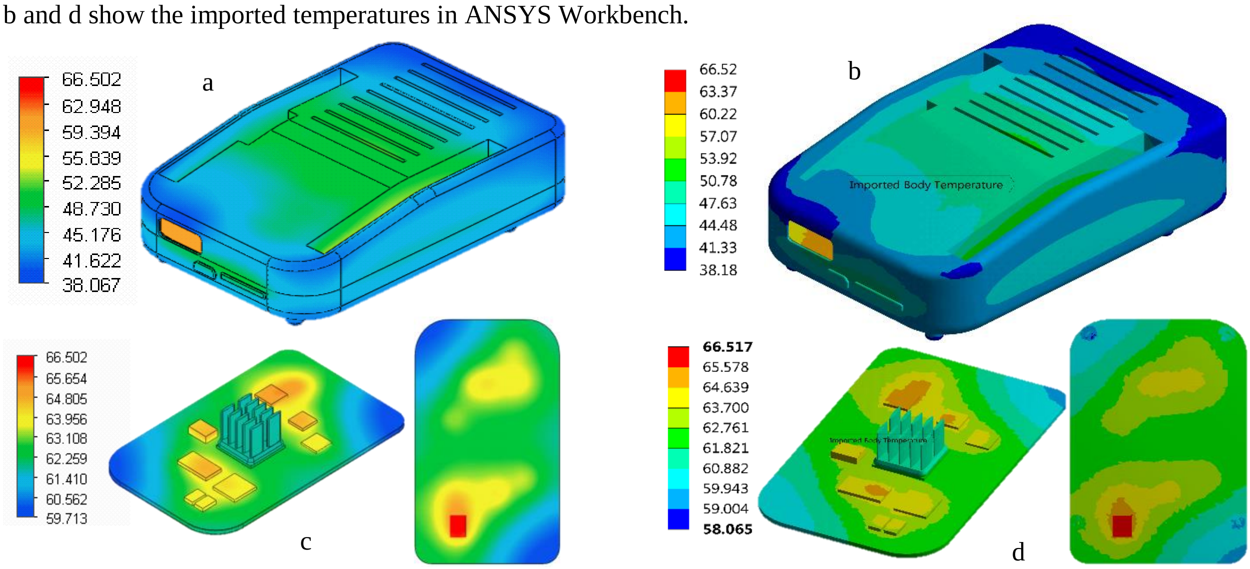

Figure 2. Temperature results

The figures show that the temperature results from Flotherm XT has been correctly mapped in ANSYS Workbench. The maximum board temperature is 66.5°C while the minimum system temperature is 38°C as expected. The mapped temperatures in ANSYS Workbench have been used to perform the thermo-mechanical analysis and the results are shown in Figure 3.

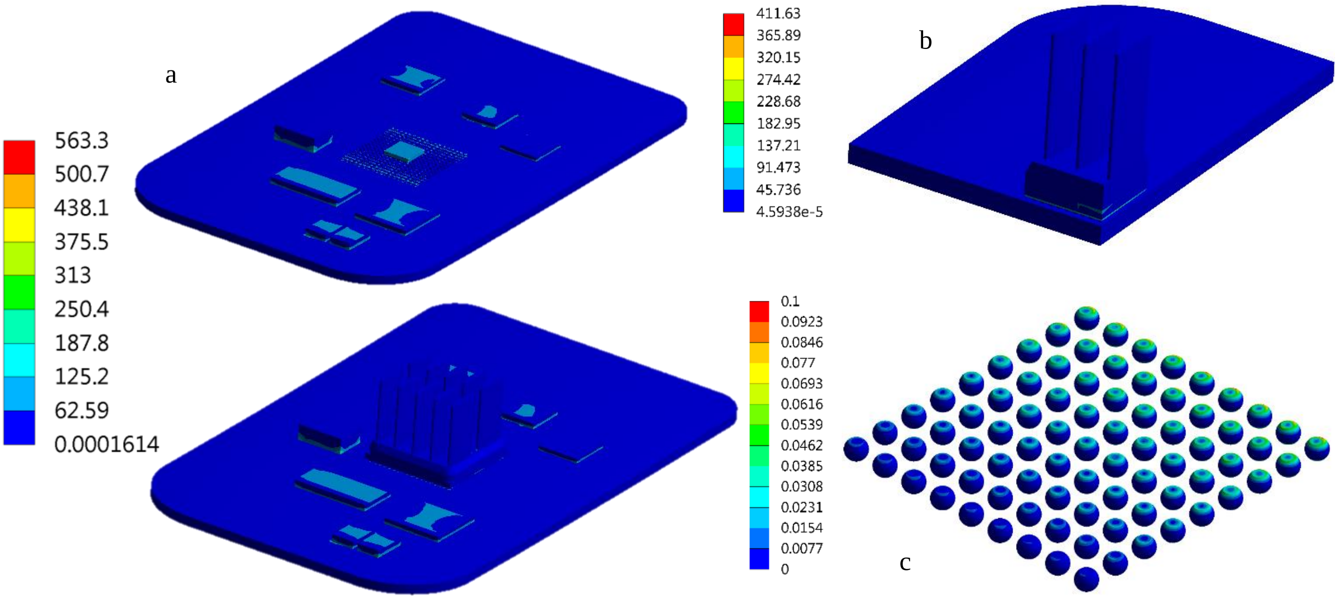

Figure 3. Beagle bone stress and strain results

Figure 3a is the board level equivalent stress distribution (von Mises stress, values shown in MPa) computed by assuming linear material properties for all the components while 3b is equivalent stress in the quarter model of the full board used to perform nonlinear analysis due to the viscoplastic nature of solder material, while 3c shows the equivalent plastic strain distribution in the BGA solder joints.

The results from Figure 3a show that the ICs have the highest stress due their high Young modulus. Solder balls directly under the flip chip are not as deformed as the solder balls around the edge of the heat sink and the flip-chip. The critical solder joints can easily be identified from such simulations to enable necessary design precautions, evaluations and optimisations to ensure optimum reliability.

Case 2: IGBT Power module under power cycle loading:

The reliability of IGBT power electronics modules is often determined through accelerated tests such as thermal and power cycling during the design stage. The power cycling tests can be simulated by using various numerical methods such as FEA and CFD to reduce testing time and cost. Power cycling is preferred, because it depicts similar field operating conditions of the power electronic systems.

Power cycling requires turning the power devices on for a period of time to allow the temperature to reach a certain value and then cooling it down to create a thermal swing, or cycle. During power cycling, the power module is cooled with a coolant which is often liquid or air. Because the power cycling requires determining the transient temperature swing under a typical electrical loading and cooling conditions, CFD analysis is best suited for this analysis which is often performed with finite volume based software (such as Flotherm XT used in this analysis) to determine temperature history under the power cycling loading. The temperature results are then used to determine the stress and strain distribution as a result of the CTE mismatch and temperature gradient using suitable software such as an FEA based software (such as ANSYS Workbench used in this analysis).

The power module with a liquid-cooled heatsink shown in Figure 4 is used in the analysis. It consists of pairs of six IGBT and diodes soldered onto a direct bonded copper (DBC) aluminium nitrate (AlN) substrate, which is soldered onto a copper baseplate. The assembly is covered with a moulding compound. A thermal interface material (thermal grease) is used to mount the assembly onto the aluminium (Al) heatsink and the whole assembly is enclosed within an Al case with nylon O-ring sealant. The coolant flow channel is in the bottom part of the Al case. All materials have been modelled as linear elastic except for the solder which is modelled as viscoplastic model and aluminium as bi-linear elastic plastic material.

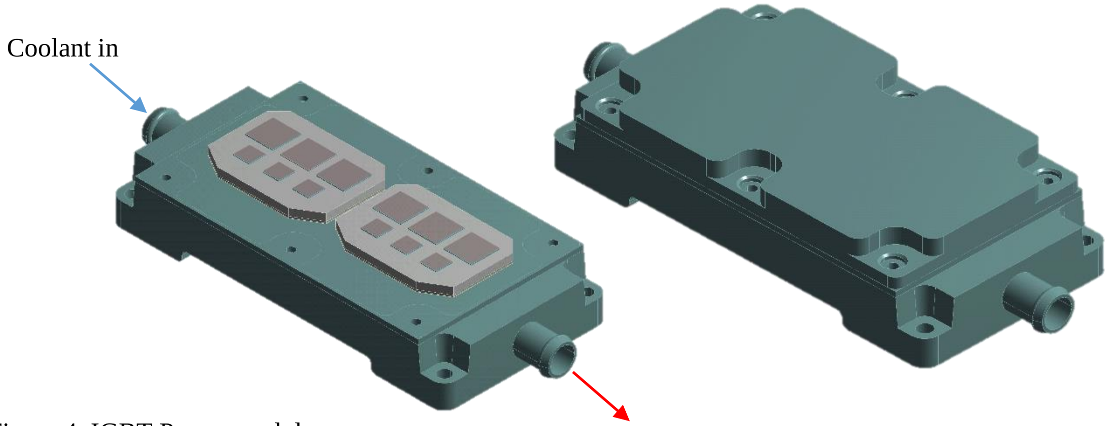

Figure 4. IGBT Power module

An average power loss of 250W and 70W is applied to the IGBT and diode power chips respectively for 3 seconds and turned off for 3 seconds giving a 6s cycle period. A water-glycol mixture coolant with 5litre/min flow rate is used. A damage metric in the form of plastic work density (inelastic strain energy density) occurring in the solder joints are compared for two loading cases: Case 1 is when power dissipation increases linearly, remains constant and then decreases linearly during the turn on time while Case 2 is when power dissipation remains constant during the turn on time. The results are shown in the Figure 5.

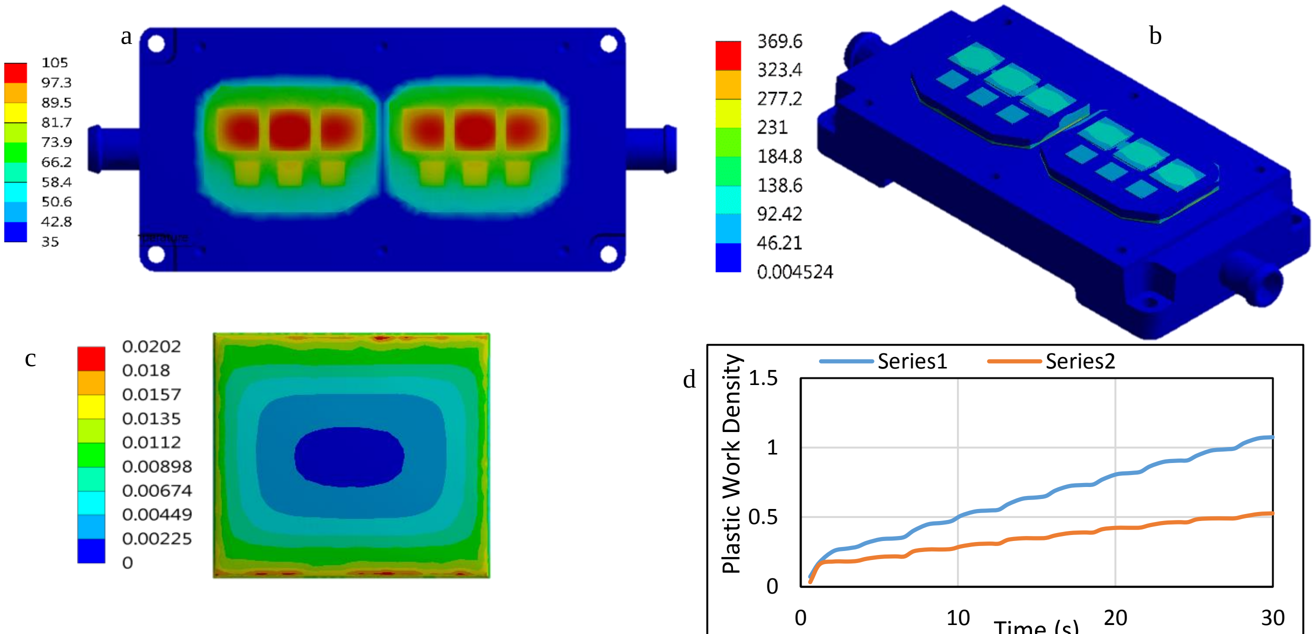

Figure 5. Power cycling results

Figure 5a is the Case 2 imported temperature in ANSYS Workbench which is consistent with the Flotherm XT temperature variation in the IGBT power module at time 2s. Figure 5b and 5c are respectively the von Mises stress (MPa) distribution in the power module and equivalent plastic strain in the power device/substrate solder at time 2s due to temperature gradient and CTE mismatch. Figure 5d shows the plastic work density increment with time for the two cases considered.

Due to the temperature swing and CTE mismatch, thermal stress and strain can be computed as shown in figure 5b as well as the plastic strain distribution in the solder joints in Figure 5c. Figure 5b shows that the high stress values are in the silicon chips (power devices) and AlN substrate due to their high Young's moduli values as expected.

From Figure 5d, the impact of loading conditions on damage accumulation can be seen. The constant power dissipation produces much more severe damage in the solder joints than the varying power dissipation values during the power cycling simulation. This type of analysis would ensure that effects and impacts of simulation assumptions can be understood.

Overall in the two simulation cases presented above, the temperature results from the Flotherm XT and the imported temperature results in ANSYS workbench correspond very well. This means that a designer with speciality in electronics thermal analysis can perform a thermal and flow analysis with FloTHERM XT and send the results to another designer/analysts with speciality in thermo-mechanical analysis who in turn performs a thermo-mechanical analysis using a different software. This speeds up the design process and improves communication between different design teams.

In conclusion, the Flotherm XT temperature results are mapped accurately in the ANSYS Workbench showing that the two software can be combined to perform a multi-physics analysis while deriving the advantages both software. The work-flow is quite straight forward with minimal user effort. With such a software combination for simulation approach, designers and analysts in multidisciplinary design fields can share high fidelity numerical simulation results, thus saving design and simulation time as well as improving robustness and flexibility.

Quote:

"In conclusion, the Flotherm XT temperature results are mapped accurately in the ANSYS Workbench showing that the two software can be combined to perform a multi-physics analysis while deriving the advantages both software. The work-flow is quite straight forward with minimal user effort."

Dr. Hua Lu, CMRG, University of Greenwich