10 TIPS FOR STREAMLINING PCB THERMAL DESIGN… A HIGH-LEVEL ‘HOW TO’ GUIDE

M E C H A N I C A L   A N A L Y S I S

W H I T E P A P E R

WHY IS PCB THERMAL DESIGN IMPORTANT?

Many aspects of a PCB's performance are determined during detailed design, e.g. making a trace a specific length for timing reasons. Timing issues are also affected by temperature differences between components. Thermal issues with the PCB design are largely 'locked in' during the component (i.e. chip package) selection and layout phases. After this point only remedial actions are possible if components are found to run too hot. We advocate a top-down approach starting at the system or enclosure level [Ref. 1] in order to understand the flow environment for the electronics, which is critical for air-cooled electronics. Assumptions made about the uniformity of the airflow in early design that subsequently proves unachievable can have a disastrous impact on the commercial viability of the product and meeting the market window.

OPTIMIZING THE THERMAL LAYOUT

The golden rule is to start early and start simple. The mechanical engineer responsible for the thermal integrity of the product should aim to provide as much useful feedback as possible to the electronic engineers to guide the design, about the thermal impact of their choices, especially during early design.

From the mechanical engineer's perspective, at the PCB level this entails helping with package selection and the best positioning of components to utilize system air flow for cooling. Inevitably both layout and package selection are driven primarily by a combination electronic performance and cost considerations, but the consequences of those choices on thermal performance should be made as clear as possible, as temperature and cooling also affect performance and cost.

1: START PRE-PLACEMENT / PRE-LAYOUT

There is a lot of work that can be done well before layout is completed within the electrical design flow. Indeed, any influence thermal considerations have on the design need to be factored in **before** this point. A lot of work can be done with a simple representation of the enclosure [Ref. 1] to provide information about the air flow profile over the board.

You can start by simply smearing the total board power over the total board surface. This will give you a temperature map that will give an indication of any hot regions that are caused by a maldistributed air flow, and the enclosure-level air flow should be optimized ahead of the PCB design. For this you can treat the board as a block with an isotropic conductivity of between $5Wm^{-1}K^{-1}$ and $10Wm^{-1}K^{-1}$. The results at this stage will be quite insensitive to the value chosen.

A word of caution – components inject heat locally into the board, so the heat flux density into the board below a component will be higher than the average for the board. As a result, the local board temperature will be higher than that predicted in the simulation, so at this stage the board temperature should not be used to try to estimate component temperatures. To do that the model needs to be refined.

If the board temperature at any point is close to the maximum component case temperature, then it is very likely that this limit will be exceeded once the component heat sources are represented discretely. This may be expected for example if one or more components are known to require a heatsink.

2: GET COMPONENT POWER (GUESSTIMATE)

For this reason it's very important to know a best guessed estimate (guesstimate) of the individual power budgets for the main heat dissipating components that will be used in the design, and the approximate size of those packages. This will allow you to describe these as footprint heat sources in your simulation, smearing the remainder of heat uniformly over the board surface.

Ahead of part research and selection, which happens at the start of the schematic capture phase of the project, the system architect will already have some idea of what the key components will be required, what will need to be positioned close to what, the size the components will be etc. For example, he may be anticipating using some components that were selected for another product, or retain components from the previous generation product.

3: USE 3D COMPONENT MODELS BEFORE PACKAGE SELECTION

It's important to try, despite the difficulty, to include some form of 3D component model in the simulation **before** the component selection is finalized. By feeding the thermal results back before this milestone is reached it's more likely that thermal performance will get considered as part of the package selection criteria. Some ICs are available in more than one package style, and not all package styles perform equally well from a thermal point of view. As a result, the need for a heatsink later on may be eliminated by appropriate package selection.

Component temperature, either in the form of a case temperature or junction temperature depending on how the manufacturer has specified the component is the key measure used to indicate whether the design is acceptable from a thermal perspective. At this stage however, we can only get a rough estimate of component temperature.

In the absence of any other information, the simplest 3D component model that can be used is a conducting block. FloTHERM includes material properties that are tailored to give a case temperature prediction for different package styles.

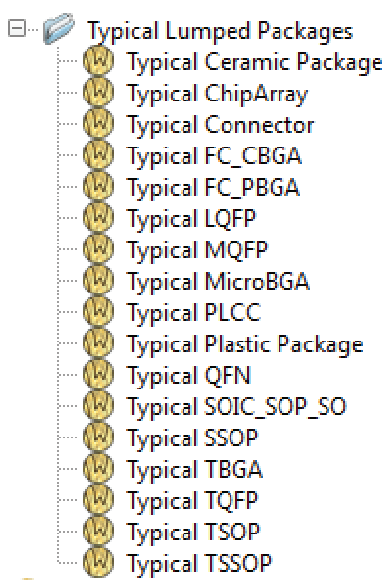

Figure 1: Lumped Package Materials in FloTHERM

For plastic components a thermal conductivity of $5Wm^{-1}K^{-1}$ to $10Wm^{-1}K^{-1}$ is recommended [Ref. 2] and $15Wm^{-1}K^{-1}$ for any ceramic components. $5Wm^{-1}K^{-1}$ will clearly give a worst-case figure for case temperature.

By representing the package body in three dimensions the effect of the component on the local air flow and correspondingly any downstream components is taken into account. Large components can shield smaller, lower profile components from cooling air, and the wake formed behind a component is a region in which the same air gets recirculated, so any components in that region are likely to be hot. One useful tip is to try to align any rectangular components so that their long side is parallel to the primary flow direction. This both reduces the overall pressure drop as the flow 'sees' less of an obstruction and produces a smaller wake, minimizing the effect on downstream components.

4: FEED BACK THERMAL RESULTS

At this stage you can start to feed information about the PCB's performance back to the PCB design team. Although the simulation is relatively coarse at this stage, the principle simulation results, being the airflow distribution over the board and the resulting board temperature map are very powerful tools that you can use to show what you have to work with in terms of available cooling air and what that may mean in terms of component temperature.

It is worth emphasizing that these nominal component case temperature values are subject to change as they are based on:

- an assumed layout,
- very rough power estimates
- uncertainty about package selection
- unknown layer stack-up and copper distribution within the PCB, and
- a preliminary heatsink size and design (if already known to be necessary)

It is nevertheless a useful start, providing both an understanding of the system performance and a model that can be refined as the design is elaborated. This model provides a very useful platform for investigating the effect of component placement on the temperature of a component and its neighbors, so adjustments can be made easily and the model re-run often in a matter of minutes not hours.

The results will give some indication as to which components, if any, will be likely to need some form of heat sink, which can be investigated next. These are also the components that need to be refined in terms of their modelling once more information is known about package selection, so this exercise helps prioritize where to invest effort in developing the thermal model.

5: SIZE HEATSINKS EARLY

For any components that may be too hot, investigate how effectively the use of a heatsink brings down the component's temperature. If the flow is mainly normal to one side of the package, a plate (or extruded) fin heatsink is likely to be most suitable. If not, then a pin fin heatsink should be considered.

FloTHERM and FloTHERM XT have a Heatsink SmartPart which can be used to parametrically define heatsink geometry. Start by making the base of the heatsink the same size as the package and investigate different numbers of fins, fin height and fin thickness. The aim is to see if the heatsink can simply be mounted on top of the package, or if a larger heatsink might be needed, which will require board real estate for the mechanical attachment (see Figure 2) as this information needs to be fed back to the PCB design team as early as possible. If so, it is essential to select an existing heatsink that provides adequate cooling performance, or design a custom heatsink, before the board can be routed, as the mechanical attachment for the heatsink may affect component placement.

Heatsinks are essentially area-extending devices, which increase convective heat transfer to the air by providing a larger surface area for the air to pass over. Heatsinks are normally made of an aluminum alloy to allow the heat to spread effectively across the base and up the fins. The base itself acts as a heat spreader and so helps to reduce the component temperature. Start by using short, widely spaced fins to minimize the resistance to the airflow and any wake caused by the heatsink as this will impact the cooling of downstream components.

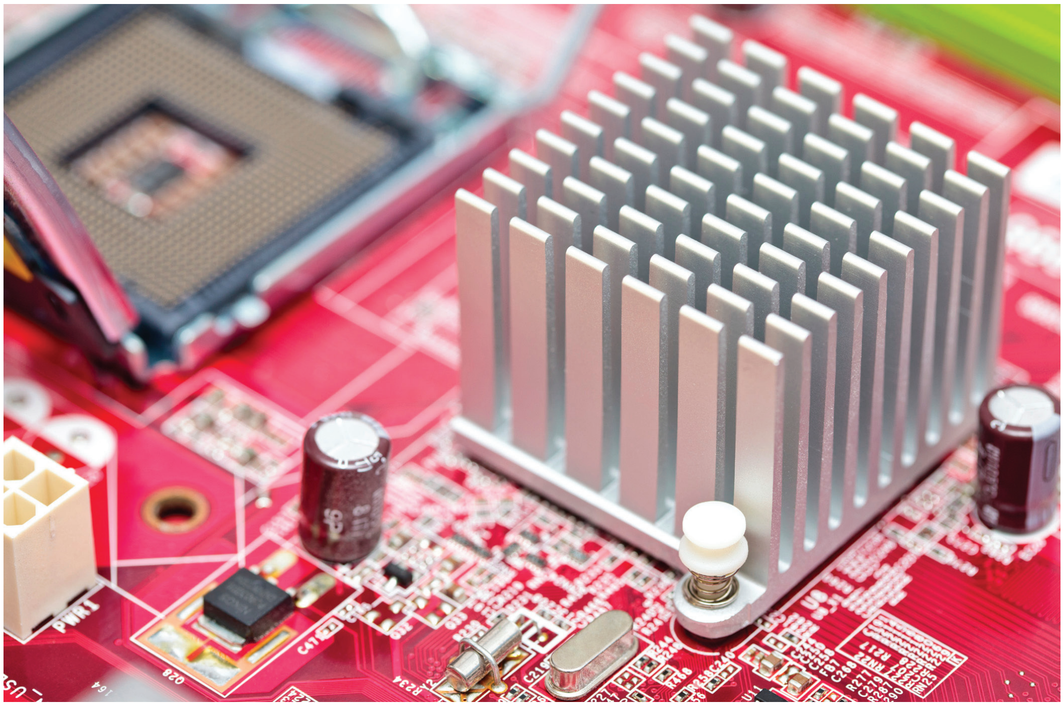

Figure 2: Heatsink extending beyond package body with retaining pins

If this shows that the component can be cooled by a relatively small component-mounted heatsink this activity can stop at this point, but will need to be revisited later.

When including a heatsink it is essential to include the thermal resistance of the thermal interface material (TIM) between the package and the heatsink. The ultimate choice will depend on many things, but a standard thermal pad having a thickness of around 0.2mm and a thermal conductivity of around 1.0 $Wm^{-1}K^{-1}$ will be conservative for early design use.

6: REPRESENT THE COMPONENT ACCURATELY

Having fed information back to the PCB design team to help with component selection and placement, guidance can also be given on the most relevant thermal metrics to use to compare the thermal performance of candidate components.

For components without a heatsink the most relevant thermal metric to compare is the junction-to-board resistance [Ref. 3]. For components that are expected to have a heatsink the junction-to-case resistance is the most relevant, as the resistance is usually defined for the face that is in contact with the heatsink [Ref. 4]. For TO type packages this face is normally soldered to the PCB. If both of these metrics are available, a JEDEC standard 2-Resistor model (see Figure 3) can be created and the thermal model re-run to get a first estimate of junction temperature [Ref. 5].

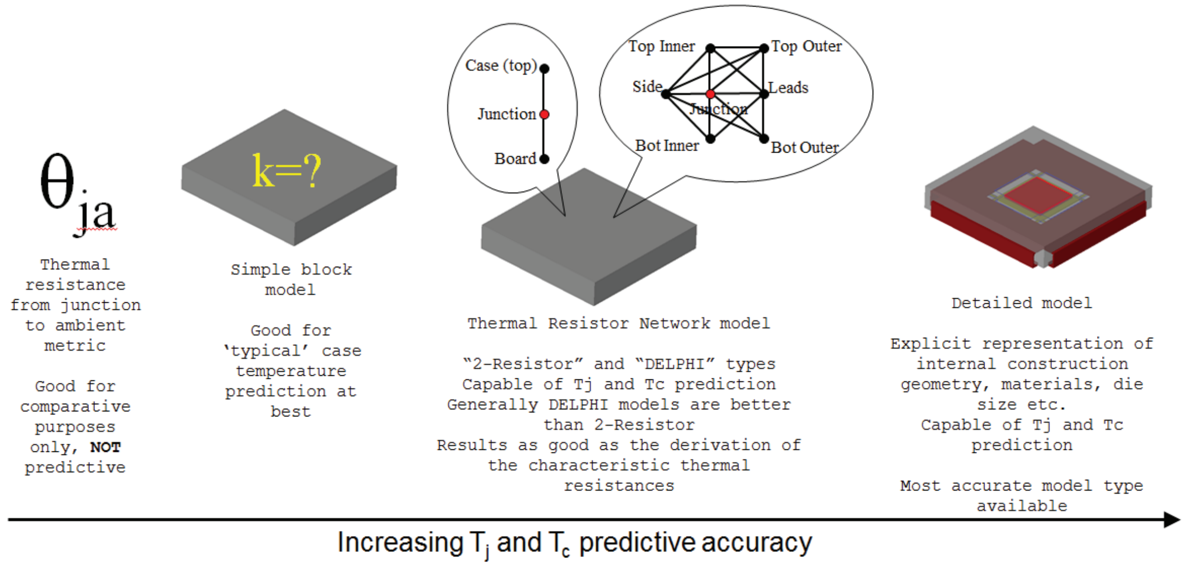

Figure 3: Fidelity Hierarchy of Package Thermal Information for Design

The next level up in terms of predictive accuracy is a DELPHI model [Ref. 6]. DELPHI models are better for heatsink selection than 2-Resistor models as the top surface is sub-divided into inner and outer regions that have different temperatures, and so can be used to initially investigate the effect of heatsink base thickness. However, for thermally-critical packages that require a heatsink the use of a detailed model is recommended.

It is also well worth doing an internet search for the components datasheet and to see if there are any FloTHERM models available and if not request a FloTHERM model from your supplier. Sometimes these are provided under a Non-disclosure Agreement (NDA). As the most widely adopted electronics cooling CFD software many of the leading IC package vendors provide thermal models for FloTHERM. This is also where FloTHERM PACK really comes into its own. Around 70% of all FloTHERM PACK users are systems integrators, who use FloTHERM PACK's JEDEC Package Wizard to generate a representative thermal model of the package just by knowing the package style, body size and number of leads. FloTHERM PACK also gives you full access to all the input data, so you can update the model as soon as more information about the package becomes available, and generates 2-Resistor, DELPHI [6], and detailed models so the component thermal model can be easily refined as the design is elaborated.

7: DON'T IGNORE THE BOARD DETAIL

The sensitivity of the results to the board thermal conductivity is also something that can be investigated once component footprint and heat source estimates are available and components modelled as 3D conducting blocks, so this activity can, and indeed should, happen in parallel with refinements to the component models.

In practice there is no single value for PCB thermal conductivity. PCBs are made up of copper and dielectric material, with the copper being about 1000 times more thermally conductive so the dielectric thermally insulates the layers from one another and insulates individual traces. In early design, well before the board has been tracked a simple isotropic conductivity value can be used and varied say between $5Wm^{-1}K^{-1}$ to $15Wm^{-1}K^{-1}$ to see how big an impact the thermal performance of the PCB has on the simulation results. This thermal representation of the board will need to be improved during detailed design.

Once the placement has been broadly defined, the next step for the PCB design team is schematic capture and electrical simulation (e.g. timing). The most useful information that can be obtained after the schematic capture, but before the board is routed, is the layer stack-up of the board.

It's important to get an estimate of how many signal, and power or ground layers the board will probably have. Traces on the surface of the PCB locally spread heat away from the package interconnect (leads or solder balls), whereas buried power and ground planes increase the in-plane thermal conductivity at the macroscopic scale.

From a thermal perspective, the contribution of these copper-containing layers on the performance of the PCB is influenced by their thickness. The most common thicknesses are 0.5Oz or 1.0Oz copper. 'Oz' denotes the weight in ounces of copper, spread evenly over a one square foot area [Ref. 7]. 1Oz is equal to 1.37mils (thousands of an inch) or 0.0347mm.

Once you have an estimate of the number of each type (signal, or power / ground) of layer in the PCB the model of the PCB can be upgraded to include each of these layers individually. Before routing an estimate needs to be made for the thickness and percentage copper coverage of each non-dielectric layer. 1Oz should be used for power and ground planes and 0.5Oz for trace layers, with an assumed copper coverage of 80% and 20% respectively. As the dielectric contributes little to the area-averaged conductivity, both in-plane and through-plane, the conductivities of these layers can therefore be taken to be 80% and 20% of the thermal conductivity of copper respectively.

The minimum thickness of the dielectric layers depends on the thickness of copper either side to allow for the difference in the coefficient for thermal expansion [Ref. 8], which then gives the total thickness of the board.

For small, high-power, low pincount packages the length scale of traces on the board are a similar order of magnitude to the package, so it is important to model these features in a similar level of geometric detail to the package before this information is available from the EDA system, for example representing the copper pad that a TO package is soldered to, and traces local to the package, when the package is modelled in detail. Similar comments apply to modelling any thermal vias below the pad used to conduct heat down to a buried ground plane.

8: IMPORT DATA FROM THE EDA SYSTEM

FloTHERM and FloTHERM XT have comprehensive EDA interfacing capabilities that allow import from all the leading EDA systems: Mentor PADS, Mentor Boardstation, Mentor Xpedition Enterprise, Cadence Allegro, and Zuken CR5000.

Importing component placement data from the EDA system ensures that placement within the thermal tool is correct, and should be reimported whenever the layout is changed. FloTHERM XT's FloEDA Bridge module allows updates to the PCB design data to be re-imported with the touch of a button, retaining all existing settings about how this data has been filtered by the user.

Detailed PCB Modeling involves importing the stack up, the routing of trace layers from the EDA system, the distribution of vias as well as the copper shapes on power and ground planes.

9: MOVE PCB THERMAL DESIGN UP THE DESIGN FLOW

In the spirit of continuous improvement, aim to move thermal design considerations up the design flow for the next project.

In part this entails paralleling up the thermal design from a mechanical perspective with the thermal design from an electrical perspective. Done well, the two approaches complement one another and can lead to the thermal design closing faster, far more reliably, and with a better outcome than if thermal design is undertaken in only one flow. Key to this is a shared understanding of what work can and should be done in each flow, which is what this document is intended to help facilitate.

10: UNDERTAKE CO-DESIGN WITH THE EDA FLOW

The ultimate goal then is thermal co-design with the EDA flow. Current densities in PCB traces and through neck-downs between different regions on power planes have increased in recent years, making Joule (or ohmic) heating an increasing issue in PCB design, affecting both the electrical and thermal performance of the board. To help electronic designers address this issue sophisticated analysis tools such as Mentor Graphics' HyperLynx Thermal and HyperLynx PI (for power integrity) complement layout and routing toolsets.

Joule heating results in additional heat sources within the PCB itself, adding to the heat dissipated by the active components. FloTHERM can import a detailed heat source map on a layer-by-layer basis to superimpose onto a detailed model of a PCB to correctly account for this.

CFD software such as FloTHERM provides the most accurate representation of the convective cooling of the PCB, plus radiative exchange with any surrounding objects. Importing heat sources resulting from Joule heating within the PCB from HyperLynx PI into FloTHERM or FloTHERM XT is recommended before the thermal design for the entire system is closed.

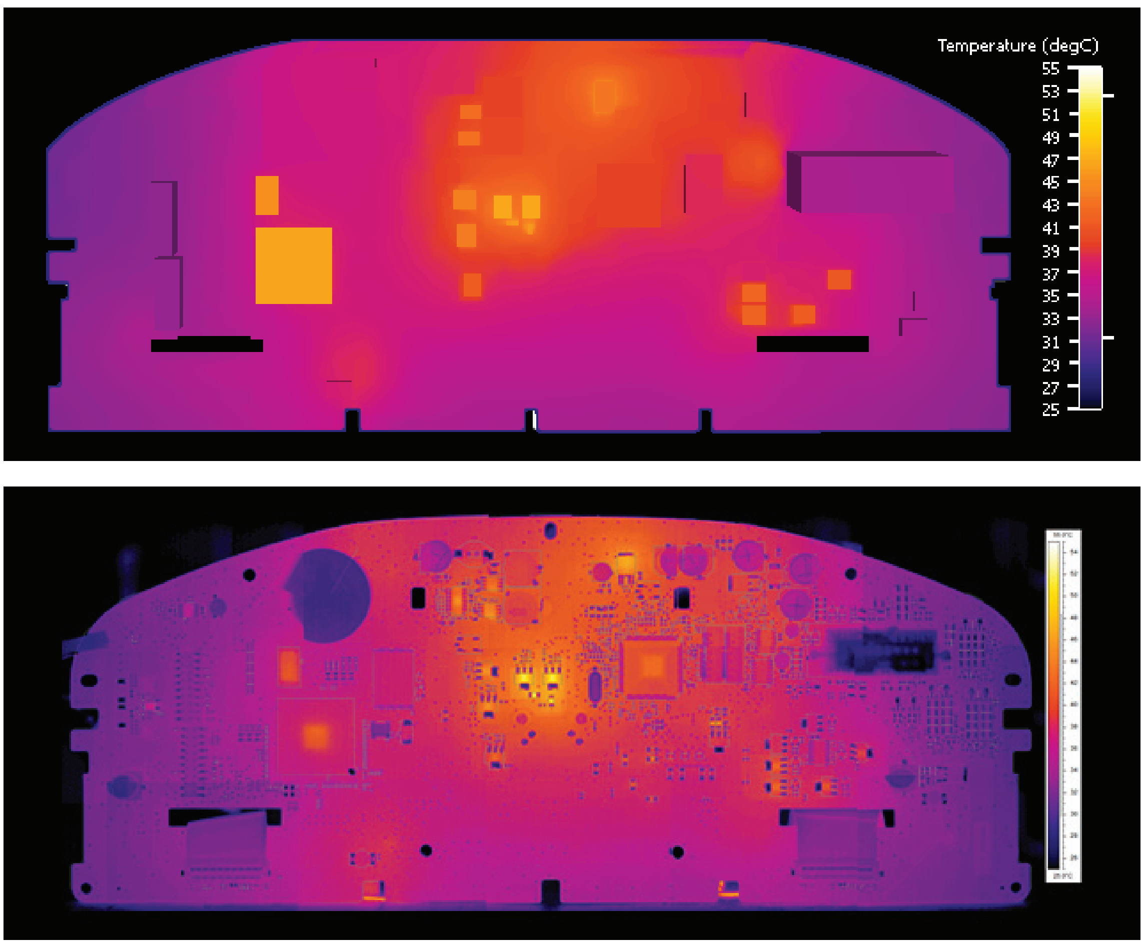

Figure 4: Comparison of CFD Result During Design With Actual PCB Measured With An IR Camera

CLOSING REMARKS

This white paper provides an overview of the key considerations in PCB Thermal Design. It is by no means exhaustive, with many details not covered. If you are responsible for PCB-level thermal design and want to know how Mentor Graphics' thermal design software can help, and what is the right product for your application, then please contact us through the Mentor Graphics Mechanical Analysis web page
www.mentor.com/mechanical

ACKNOWLEDGEMENTS

Dr John Parry, CEng, CITP, MBCS, MIEEE
Byron Blackmore

REFERENCES

1. **12 Key Considerations in Enclosure Thermal Design… A High-Level 'How To' Guide, Mentor Graphics White Paper**
2. Tony Kordyban (1994) Estimating the Influence of PCB and Component Thermal Conductivity on Component Temperatures in Natural Convection, Third International FloTHERM User Conference, September 1994, Guildford UK.
3. Integrated Circuit Thermal Test Method Environmental Conditions — Junction-to-Board. JEDEC JESD51-8, October 1999.
4. Transient Dual Interface Test Method for the Measurement of the Thermal Resistance Junction to Case of Semiconductor Devices with Heat Flow Through a Single Path. JEDEC Standard JESD51-14, November 2010
5. Two-Resistor Compact Thermal Model Guideline. JEDEC Guideline JESD15-3, July 2008
6. DELPHI Compact Thermal Model Guideline. JEDEC Guideline JESD15-4, October 2008
7. **https://en.wikipedia.org/wiki/Printed_circuit_board**
8. **http://www.pcbuniverse.com/pcbu-tech-tips.php?a=4**

Corporate Headquarters
Mentor Graphics Corporation
8005 SW Boeckman Road
Wilsonville, OR 97070-7777
Phone: 503.685.7000
Fax: 503.685.1204

Sales and Product Information
Phone: 800.547.3000
sales_info@mentor.com

Visit www.mentor.com/company/office_locations/ for the list of Mechanical Analysis Division Offices

MGC 01-14 TECH11770