IMPROVEMENT OF A STEADY STATE METHOD OF THERMAL INTERFACE MATERIAL CHARACTERIZATION BY USE OF A THREE DIMENSIONAL FEA SIMULATION IN COMSOL MULTIPHYSICS

BENJAMIN SPONAGLE AND DOMINIC GROULX
Dalhousie University, Nova Scotia, Canada

Excerpt from the Proceedings of the 2012 COMSOL Conference in Boston
• The Lab of Applied Multiphase Thermal Engineering at Dalhousie University has a contract with Raytheon to work on the characterization of thermal interface materials (TIMs).

• The first step in the project was to build and test a steady state characterization device.

• The goal of the work presented here was to create an FEA simulation of that test device which could be used to improve the accuracy of the experiment.
BACKGROUND

• What are Thermal Interface Materials?
• Materials which are designed to increase the thermal conductance of an interface between two surfaces.
• A common application is to reduce resistance of the conduction path from a microchip to a heat sink.
BACKGROUND

- Problem Statement
- Background ✓
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks

Illustration of contact resistance [1]
BACKGROUND

- The performance of a TIM is a function of:
  - Effective thermal conductivity
  - Ability to conform to surface features
  - Thickness of the TIM layer
- It is not possible to characterize the performance of a TIM with a single property such as bulk thermal conductivity.
- Roughness and surface flatness are important parameters.
- Performance will vary with both clamping pressure and TIM temperature.
- We must measure its performance while it is in an interface.
An apparatus was designed to allow for steady state testing.

Steady state testing was based on the ASTM standard includes a guard heater, insulating sheath six RTD temperature sensors.

The goal of the steady state test is to setup a one dimensional heat flow through the TIM sample.
METHODOLOGY

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks
METHODOLOGY

\[ \theta = QA^{-1}(T_H - T_C)^{-1} \]

\[ \Delta T_{H\rightarrow C} = \Delta T_{3\rightarrow 4} - d_{3\rightarrow H} \left( \frac{dT}{dz} \right)_{hot} - d_{C\rightarrow 4} \left( \frac{dT}{dz} \right)_{cold} \]

\[ \left( \frac{dT}{dz} \right)_{hot} = \frac{\Delta T_{1\rightarrow 3}}{d_{1\rightarrow 3}} \]
\[ \left( \frac{dT}{dz} \right)_{cold} = \frac{\Delta T_{4\rightarrow 5}}{d_{4\rightarrow 5}} \]

\[ Q = kA \left[ \left( \frac{dT}{dz} \right)_{hot} + \left( \frac{dT}{dz} \right)_{cold} \right] \]

\[ \theta = \left[ \frac{k}{2} \left( \left( \frac{dT}{dz} \right)_{hot} + \left( \frac{dT}{dz} \right)_{cold} \right) \right] \left[ \Delta T_{3\rightarrow 4} - d_{3\rightarrow H} \left( \frac{dT}{dz} \right)_{hot} - d_{C\rightarrow 4} \left( \frac{dT}{dz} \right)_{cold} \right]^{-1} \]

\( \theta \) = Thermal Conductance
\( Q \) = Heat Flow
\( k \) = Thermal conductivity of the meter bar
\( A \) = Cross sectional area of the meter bar

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks
METHODOLOGY

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks
COMSOL MODEL

- Problem Statement
- Background
- Methodology
  - COMSOL Model
- Mesh
- Results
- Concluding Remarks
COMSOL MODEL

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks
COMSOL MODEL

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg/m³)</th>
<th>Thermal Conductivity (W/mK)</th>
<th>Specific Heat (J/kgK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Al 6061 T6</td>
<td>2700</td>
<td>167</td>
<td>900</td>
</tr>
<tr>
<td>Superwool 607</td>
<td>335</td>
<td>0.06</td>
<td>0.243</td>
</tr>
<tr>
<td>Macor</td>
<td>2520</td>
<td>1.46</td>
<td>790</td>
</tr>
<tr>
<td>Air</td>
<td>COMSOL Materials Database</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
COMSOL MODEL

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks

\[ K_{TIM\text{domain}} = d_{TIM\text{domain}} \theta \]
Problem Statement
Background
Methodology
COMSOL Model
✓ Mesh
Results
Concluding Remarks

- Hex elements through most of the model, tetrahedral elements in the heat exchanger
- Maximum Element Size: 0.002 m
- 4 elements through the insulation layer: 0.0064 m
MESH

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks

![Graph showing temperature versus maximum element size](image-url)

- RTD1
- RTD2
- RTD3
- RTD4
- RTD5
- RTD6

Temperature (°C)

Maximum Element Size (m)
MESH

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh
- Results
- Concluding Remarks
RESULTS

- To do an initial test of the model the simplest experimental dataset was chosen: No TIM at 0.50 Mpa (73 psi)
- This is a well understood case with few heat losses. Model is expected to match the one dimensional case well.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h_{\text{lateral heat loss}}$</td>
<td>2.0 W/m$^2$K</td>
</tr>
<tr>
<td>$h_{\text{heatsink}}$</td>
<td>36.1 W/m$^2$K</td>
</tr>
<tr>
<td>$q_{\text{top heat loss}}$</td>
<td>1.1 W</td>
</tr>
<tr>
<td>$\theta$</td>
<td>?</td>
</tr>
</tbody>
</table>
RESULTS

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh

✓ Results
- Concluding Remarks
RESULTS

- Problem Statement
- Background
- Methodology
- COMSOL Model
- Mesh

✓ Results
- Concluding Remarks

\[ \theta_{\text{model}} = 0.5 \text{ W/cm}^2\text{K} \quad \theta_{\text{exp}} = 0.49 \text{ W/cm}^2\text{K} \]
CONCLUDING REMARKS

• Initial comparisons with experimental data indicate that the FEA simulation works well.

• Further comparing the model to more data sets, especially higher temperature tests where heat losses will be more significant, is the next step.

• A sensitivity analysis on the FEA model would also be useful.
REFERENCES