Considerations for Predicting Thermal Contact Resistance in ANSYS.

Mary Kathryn Thompson, Ph.D.
Department of Civil and Environmental Engineering, KAIST, Daejeon, Korea

John M. Thompson, Ph.D.
Consulting Engineer

E-mail : mkt@kaist.edu

In many finite element models, thermal contact resistance is either neglected or included only as an intrinsic characteristic of the system. This work presents a discussion of the issues associated with predicting thermal contact resistance and methods for overcoming some of the difficulties associated with doing so in ANSYS.

Keywords: thermal contact resistance, surface roughness

1. INTRODUCTION

In many finite element models, thermal contact resistance is either neglected or included only as an intrinsic characteristic of the system. This work presents a discussion of the issues associated with predicting thermal contact resistance and methods for overcoming some of the difficulties associated with doing so in ANSYS.

2. THERMAL CONTACT RESISTANCE

No material is a perfect thermal conductor and thermal losses, observed by temperature drops, occur as heat travels through real systems. This phenomenon is often modeled using an electrical analogy to compare the voltage drop across an electrical circuit to the temperature drop across the interface. For the electrical system, Ohm's Law states:

$$V = IR \qquad (1)$$

where V is the voltage drop across a resistor, I is the current flowing through the circuit, and R is the electrical resistance of the resistor. In a thermal system, the analogy to Ohm's Law has a similar form and is stated as:

$$\overline{\Delta T} = QR \qquad (2)$$

where $\overline{\Delta T}$ is the average temperature drop across the thermal body, Q is the heat flowing through that body, and R is the thermal resistance of the body. When the temperature drop occurs at the interface between two bodies or materials, the phenomenon is referred to as thermal contact resistance (TCR). Since all real thermal systems are three dimensional, this analogy produces a quasi-1D (averaged) approximation to a 3D problem.

Thermal contact resistance is a function of surface geometry, properties of the materials that are interacting at the surface(s), properties of materials (air, thermal grease, etc.) in the interface if present, applied mechanical loads, and applied thermal loads. Surface coatings, surface chemistry, and surface damage or defects can also influence thermal contact resistance.

3. THERMAL CONTACT CAPABILITIES IN ANSYS

The ANSYS surface-to-surface and node-to-surface contact elements (CONTAC 171-175) have extensive built-in capabilities for modeling thermal contact including thermal conduction, free surface and surface-to-surface thermal convection, free surface and surface-to-surface thermal radiation, and heat generation due to frictional dissipation [1].

The mode(s) of heat transfer across the interface are determined by the status of the contact elements. If the element is in closed contact, then heat is transferred via thermal conduction. If the element is in closed contact and sliding in a transient analysis, then heat is generated by friction if friction is defined for the contacting pair. If the element is in near-field contact as defined by the pinball size, then heat can be transferred by convection and radiation if convection and radiation properties are defined for the contact pair. And, if the element is in free-surface contact (open contact), then heat can be transferred to the ambient environment by convection and radiation [2].

4. THERMAL CONTACT RESISTANCE IN ANSYS

There are several challenges to predicting thermal contact resistance in ANSYS. These include the multi-physics and multi-scale nature of thermal contact problems, issues associated with determining the TCC value to use, accurately modeling surface geometry, and post-processing thermal contact resistance problems.

4.1. Multi-Physics Considerations

Thermal contact problems are coupled thermal/ structural problems. The structural deformation of the contacting surfaces determines the nature of the contact at the interface, which in turn determines the distribution, nature and magnitude of heat transfer across the interface. Simple systems can be solved sequentially with the structural problems solved first and the thermal problem solved second. However, more complex problems, which involve thermal expansion, temperature dependent material properties or other coupled phenomena, require either a coupled solution or an iterative solution. Multi-physics elements, like SOLID5 and SOLID98, can ~~must~~ be used for these types of problems to directly couple the thermal and structural parts of the analysis and reduce the overall computational effort.

4.2. Thermal Contact Resistance and Thermal Contact Conductance

In ANSYS, the resistance to solid/solid thermal conduction per unit area at the interface is included as a user supplied real constant value of thermal contact conductance (TCC) [W/m^{2}K]. The relationship between TCC, heat flux and temperature is given by:

$$q = TCC\ (T_{hot} - T_{cold}) \qquad (3)$$

where q is the heat flux per unit area [W/m^2], T_{ho}t [K] is the local temperature on the hot surface and T_{cold} [K] is the local temperature on the cold surface [3]. Local temperatures are the integration point temperatures of the elements defining the contact pair. By including TCC as a real constant, it is assumed that thermal contact conductance is known, uniform, and solution independent. This also assumes that the surface is relatively smooth and that the overall TCC value provides an adequate definition of the heat transfer between the two surfaces.

At the micro scale, the TCC real constant value truly represents thermal contact conductance and its value is the inverse of thermal boundary resistance (TBR). This value can be obtained from experiments or from the literature. Our current understanding of TBR is limited and values of TBR are very difficult to predict. At this time, it is reasonable for TBR to be modeled as uniform and solution independent.

At the macro scale, however, the TCC value represents the thermal contact resistance based upon what is happening at the micro scale interface. For some systems, the value of TCC may not have a large influence on the system and an appropriate value may be determined experimentally or obtained from the literature. However, for other systems, the value of TCC will depend on the nature of thermal contact, including geometry and heat transfer modes, at the macro and micro scales and so will not be constant or solution independent.

It is possible to predict appropriate and solution dependent values for thermal contact resistance using ANSYS by using a multi-scale iterative approach [4]. However, this requires the ability to accurately model micro scale surface geometry, and the ability to calculate gap dependent thermal conduction.

For non-uniform values of TCC, multiple models at the micro scale will have to be solved and a table with the input parameters (contact pressure, applied thermal loads, etc.) and the resulting TCC value will be constructed. Each contact element in the macro scale model can be assigned its own real constant set whose value is based on the micro scale results table.

5. INCORPORATING MICRO SCALE SURFACE GEOMETRY

Historically, it was assumed that micro scale surface geometry could not be included in numerical contact models because of the computational resources required to solve the model [5-6]. While there are still limitations for model size today, they are no longer prohibitive and a method has been developed to incorporate surface measurement data into ANSYS models.

To import surface data into ANSYS:

1. Measure the surface geometry
2. Prepare the surface data for export
3. Export the surface data into a portable file format
4. Convert the surface data into an ANSYS array
5. Import the array into ANSYS as an input file
6. Operate on imported surface array (if desired)

The micro scale surface geometry can be measured using any method desired as long as the data can be recorded digitally and exported in a plain text format.

Surface data sets are often missing individual data points (referred to as data drop out points) and may contain other measurement artifacts such as planar surface form due to the orientation of the sample relative to the measurement device. Some surface metrology software includes tools to replace missing data and filter the data set. It is also possible to repair the surface data set using APDL once it has been imported into ANSYS, but it is easier to do this before exporting the data.

Once prepared, the surface data should be exported to a portable file format like .txt, .raw. or .xyz. For this work, the .xyz file format, which lists the x, y and z coordinate for each measured data point in a three column plain text file, is most convenient.

The surface data can then be converted into a 2D ANSYS array using a spreadsheet program or scripting language. Figure 1 shows a sample data set in XYZ format which has been converted to an ANSYS array. A custom translator program written by Karta Khalsa of Zygo, Inc. is also available to convert data in the native Zygo file format (.dat) to an ANSYS array.

	numrow = 10 numcol = 1 *dim,surfdata,array,numrow,numcol
0 0 10.1186	surfdata (1 , 1) = 10.1186
1 0 10.1154	surfdata (2 , 1) = 10.1154
2 0 10.1168	surfdata (3 , 1) = 10.1168
3 0 10.1023	surfdata (4 , 1) = 10.1023
4 0 10.1145	surfdata (5 , 1) = 10.1145
5 0 10.1192	surfdata (6 , 1) = 10.1192
6 0 10.12	surfdata (7 , 1) = 10.12
7 0 10.1197	surfdata (8 , 1) = 10.1197
8 0 10.1205	surfdata (9 , 1) = 10.1205
9 0 10.1174	surfdata (10 , 1) = 10.1174
10 0 10.1114	surfdata (11 , 1) = 10.1114

Fig. 1. Sample Surface Data: XYZ Format (left) and Array Format (right)

Once the surface data has been imported into ANSYS, the user may wish to perform additional operations on the data set including re-centering the data about the average surface height value, scaling the surface height values, and adding or subtracting other data sets to represent machining or polishing operations on the surface.

Finally, the surface geometry can be created by either creating solid model geometry using the ANSYS native solid modeler or by modifying the finite element model [7-8].

Figures 2 and 3 show measured and imported surface geometry from a lapped nickel sample with a 2 micro inch average roughness. The two figures are mirror images of each other due to differences in the display and measurement coordinate systems of the surface metrology equipment used. Major surface features are outlined for clarity.

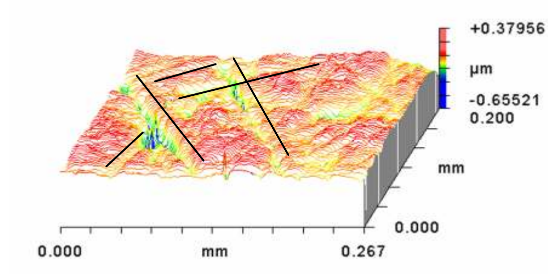

Figure 2. Measured Surface Geometry

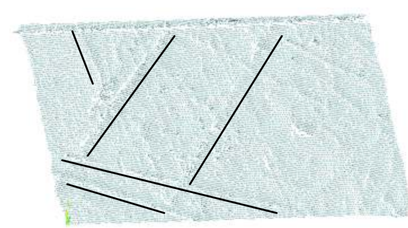

Figure 3. Imported Surface Geometry in ANSYS

6. IMPLEMENTING GAP DEPENDENT CONDUCTION IN ANSYS

As of version 11.0, gap dependent thermal conduction is not available in ANSYS. It is expected to be available in the near future. Until then, it is possible to model micro scale gap dependent thermal conduction in models that do not consider thermal convection in the gap by using the existing thermal convection capabilities. The basic procedure for implementing gap dependent thermal conduction is as follows:

1. Create and solve the thermal/structural model. Do not exit solution
2. Retrieve the contact gap for contacts in near-field contact
3. For all elements in near-field contact, apply a convection coefficient equal to the thermal conductivity of the gap material (air, thermal grease, etc.) divided by the gap length.
4. Re-solve the thermal/structural model

The first solution predicts behavior of the system with no gap dependent thermal conduction (i.e. in a vacuum). Since the nodal and element results for the last solution are available in the results section of the ANSYS database, these results may be retrieved and used to update the model parameters, such as real constant values or applied loads, before another solution is attempted. Next, a second solution is performed to predict the behavior of the system with an interstitial material. This procedure can be repeated to modify the model to obtain results for different interstitial material if desired. Sample APDL code for retrieving solution data, such as gap lengths, and modifying the surface load values to apply the convection coefficient is shown below:

```
allsel,all

*get,emax,elem,,num,max
*dim,emask,array,emax
*dim,egap,array,emax,4
*dim,ehtc,array,emax,4

esel,s,ename,,173,174
*get,ecnt,elem,,count
*vget,emask(1),elem,1,esel
cm,econt,elem

 cmsel,s,econt
   enxt=0
   *do,ijk,1,ecnt
      enxt=elnext(enxt)
      *do,jkl,1,4
         *get,epene,elem,enxt,nmisc,jkl+8
         *if,epene,ge,0,then
            egap(enxt,jkl)=0
            ehtc(enxt,jkl)=0
         *else
            epene=abs(epene)
              *if,epene,le,0.3,then
                   epene=0.3
            *endif
            egap(enxt,jkl)=epene
            ehtc(enxt,jkl)=ekxx/epene
         *endif
      *enddo

sfe,enxt,1,conv,1,ehtc(enxt,1),ehtc(enxt,2),ehtc(enxt,3),ehtc(enxt,4)
      sfe,enxt,1,conv,2,0,0,0,0
   *enddo

sfe,enxt,1,conv,1,ehtc(enxt,1),ehtc(enxt,2),ehtc(enxt,3),ehtc(enxt,4)
   sfe,enxt,1,conv,2,0,0,0,0
*enddo
```

In this example, ekxx is the thermal conductivity of the material in the interface. All other parameters are working variables.

7. POST PROCESSING THERMAL CONTACT RESISTANCE MODELS

Thermal resistances, including thermal contact resistance, are averaged values which represent the thermal losses along a 1D path. However, the results of finite element models are based on the nodal and element behavior and display the detailed behavior of the system.

Consider the thermal behavior of a symmetric model of a bolted plate system in contact. The top plate of the system (body 1) is modeled as a nominally flat plate. The lower face of the top plate has imported surface form and is in contact with the second body. The second body of the system (body 2) is a perfectly flat plate. The temperatures for the outer surfaces (11 and 22) and for the inner surfaces (12 and 21) are shown in Figure 4. The top plot (T_{11}) shows the temperature distribution on the top surface of the top plate, the second plot (T_{12}) shows the temperature distribution on the bottom surface of the top plate, the third plot (T_{21}) shows the temperature distribution of the top surface of the bottom plate and the bottom plot (T_{22}) shows the uniform temperature of the bottom surface of the bottom plate. The figure also shows the thermal circuit that describes the system. The thermal resistance through the thickness of body 1 is given by R_1. The thermal resistance through the thickness of body 2 is given by R_2. The thermal contact resistance is given $R_{contact}$ and the total thermal resistance for the system is given by the sum of the three resistors.

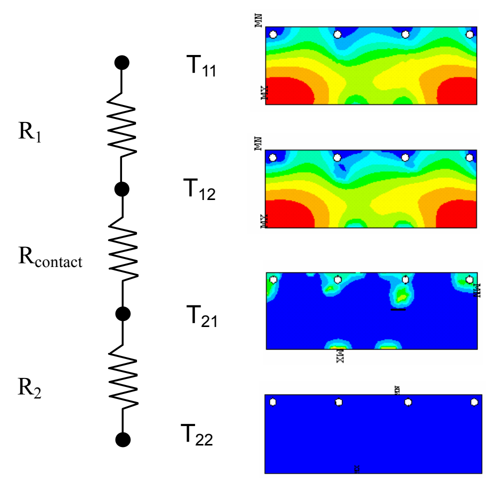

Figure 4. Thermal Circuit and Temperature Plots for a Bolted Plate System

The temperatures for three of the four surfaces are clearly non-uniform. (The fourth is perfectly uniform due to a constant temperature boundary condition.) So the heat flux across the surface and thus the thermal contact resistance will be non-uniform. But local contact resistance is rarely considered so all values must be averaged before a meaningful value of thermal contact resistance can be obtained.

To calculate the thermal contact resistance across an interface:

1. Calculate the average heat flux across the interface
2. Calculate the surface area of the contact surface

3. Multiply the average contact flux by the contact surface area to obtain the total heat flow across the interface
4. Calculate the average temperature for each side of the interface
5. Calculate the difference between the two average temperatures to obtain the average temperature drop across the interface
6. Divide the average temperature drop by the total heat flow across the interface to obtain the thermal contact resistance.

The procedures to calculate the average heat flux, total surface area, and average temperatures require significant vector operations in APDL. Macro command files were written to calculate these values to facilitate this work.

8. CONCLUSIONS

Several considerations for the prediction of thermal contact resistance, including the multi-scale and multi-physics nature of thermal contact resistance and the determination of a representative value of TCC were discussed. Methods for the importation of measured surface geometry, the implementation of gap dependent conductance and for calculating thermal contact resistance from values retrieved in the post processor were also presented.

ACKNOWLEDGMENTS

The author would like to thank Prof. Alexander H. Slocum for his advice, support and many conversations about this work; Karta Khalsa from Zygo, Inc. for his collaboration in developing the surface data translation tool used in this work; ANSYS, Inc. for donating the software used in this work; and Berk Yesin and ABB, Ltd. for their support of this work.

REFERENCES

[1] ANSYS Contact Technology Guide Section 6.1
[2] ANSYS Contact Technology Guide Section 6.1.1
[3] ANSYS Contact Technology Guide Section 6.1.4.1.
[4] Thompson, M. K., 2007, "A Multi-Scale Iterative Approach for Finite Element Modeling of Thermal Contact Resistance," Thesis (Ph.D.) Massachusetts Institute of Technology, pp. 31-43.
[5] Peng, W. and Bhushan, B. "A numerical three-dimensional model for the contact of layered elastic/plastic solids with rough surfaces by variational principle." Trans. AMSE, J. Tribology 123 (2002) 330-342.
[6] Marchand, A. S. and Raynaud, M. "Numerical Determination of Thermal Contact Resistance for Nonisothermal Forging Processes." Trans. ASME, J. Heat Transfer 122 (2000) 776-784.
[7] Thompson, M.K. "Methods for the Generation of Rough Surfaces in ANSYS." Proceedings of the 2006 International ANSYS Users Conference & Exhibition, Pittsburgh, PA. 16 pages.
[8] Thompson, M. K., 2007, "A Multi-Scale Iterative Approach for Finite Element Modeling of Thermal Contact Resistance," Thesis (Ph.D.) Massachusetts Institute of Technology, pp. 25-27.