ANALYSIS OF THERMAL STRESSES CONTACT PROBLEM OF FUNCTIONAL MATERIAL INVOLVING FRICTIONAL HEATING WITH AND WITHOUT THERMAL EFFECTS

[1]ANJANI KUMAR SINHA, [2]A. JOHN RAJAN, [3]KUMAR YOGEESH.D, [4]ERIKI ANANDA K. AR BABU

[1,3,4]Senior Lecturer, Dept. of Mech. Eng., Nilai University, BBN, Putra Nilai,71800, N.S, Malaysia
[2]Prof., Dept. of Mech. Eng., Sathyabama University, Rajeeve Gandhi Salai, 600119, Chennai, India
[5]Ph-D Research Scholar, Dept. of Mech. Eng., National Institute of Technology, Warangal, 506004, India
E-mail: [1]anjani_184@yahoo.com, [2]ajohnrajan@gmail.com, [3]dkyogeesh@gmail.com, [4]erikiananda@yahoo.co.in, [5]arbabu.1973@gmail.com

Abstract- The two-dimensional thermal effect of sliding frictional contact of material thermal stresses is investigated in this paper. Products and machinery with moving components nearly always include parts which experience frictional forces. Often, only the structural contributions to the stresses and deformation of the parts are analyzed. Frictional heating, however, creates a conduction thermal gradient along the parts, which adds to the overall stresses and deformation. Models have been developed to analyze the thermal gradients due to frictional heating for simple geometry1. We intend to develop a frictional contact model which accounts for both the structural and thermal effects on stresses and deformation of our geometry and will verify our model through a combination of linear thermal elasticity and a simplified analytical thermal conduction analysis. The purpose of this complex model is to give a more realistic simulation of the contact surface. This paper will take an in-depth look at how the frictional heat generation is incorporated into commercial solver ANSYS and compare this to an analytical solution. To accomplish this task simple practical model of frictional problem will be used for both the ANSYS simulation and the analytical solution. To aid in verification of the two solutions, a loosely coupled MATLAB code will be used on the same geometry with reasonable approximations. With this verification we hope to show how to implement frictional heating in other model, and whether it is even important to account for the frictional heating.

Keywords- Frictional Heat Partitioning, Boundary Layer, Sticking Contact, Hot Spots.

I. INTRODUCTION

The two-dimensional model and methodology developed may be used to predefine the heating partitioning factor for thermoelastic analysis in a sequential multi-physics workflow. A numerical model has been developed to study the transient thermoelastic behavior of the block sliding across two fixed walls. It is shown that the contact pressure increases over time as the temperature increases. Both temperature and pressure vary over the contact area which can explain the hot spot movement on the sliding block.

Temperature fluctuations and thermoelastic instability, and consequently hot spot evaluations between two bodies over wall, are likely to occur due to contact conditions such as pressure distribution and heat partitioning. To compare thermally and seismically induced sliding mechanisms of blocks that is separated from slide along a frictional interface. Effects of the sliding friction coefficient and are analyzed numerically by using an Arbitrary Lagrangian Eulerian Finite Element technique, analytical model are developed allowing us to interpret the numerical data and to make them more distributions along the interface are analyzed for stresses, temperature and sliding velocities. The maximum surface temperature increases significantly friction coefficient increase because a greater friction coefficient leads to much more heat flux on the contact surface and the distribution of the thermoelastic contact stress can be altered by adjusting the gradient of thermoelastic parameters of the coating would have potential applications in improving the resistance to thermoelastic contact damage.

II. SIMULATION OF GEOMETRY MODEL

A. Symmetry and 2D Analysis of model

Consider the transient case of a block sliding across two fixed walls. How will the temperature changes due to the frictional forces affect the strains and stresses? Our symmetrical 2D model as shown in Figure.1, and the constraints are as followed: A-Constant displacement in the negative Y- direction; B-This is essential boundary condition with zero X and Y displacement and a set temperature; C-This is a tabular displacement boundary condition in the X-direction



Figure 1: 2D geometry model

A-boundary condition; is an essential boundary condition due to the fixed displacement. This is necessary to induce the compounding effect of the growing thermal expansion with the increasing heat generation. The X-direction in this constraint is free, which represents the symmetry of the model.

B-boundary condition; fixes the lower part, creating a stationary part for the frictional interaction. This boundary condition also sets an essential temperature boundary condition which is vital to solving the thermal problem.

C-boundary condition; is a tabular displacement that moves the top part in the x-direction at a constant velocity, thus covering an equal distance with each time step. This boundary condition creates the time dependence allowing for the growing thermal expansion forces and frictional heating.

B. Finite Element Approach

i. Pre Processing and Meshing: Ansys helps to build a complete finite element mode, including physical and material properties, loads and boundary conditions, and analysis the various behaviors of mechanical components and structure. Preprocessing comprises of building, meshing and loading the model created. Ansys offers a complete set of tools for automatic mesh generation including mapped meshing and free meshing can access geometric information in the form of point, curves and surface. With all parts of model defined, nodes, elements, restraints and loads, the analysis part of the model is ready to begin. The system can determine approximate value of stress, deflections, temperatures, pressures and vibrations nodes. An analysis requires Nodal point, Elements connecting the nodal points, Material and physical properties, Boundary conditions which consist of loads and constraint, Analysis option: how the problem will evaluated. After creation of solid modeling the model has converted to FEM model, i.e. generating of nodes and elements: Set element attributes, Set mesh control, Generate the mesh. Before generating the mesh, definition of appropriate element attributes needed. Loading conditions as shown in Figure.2.

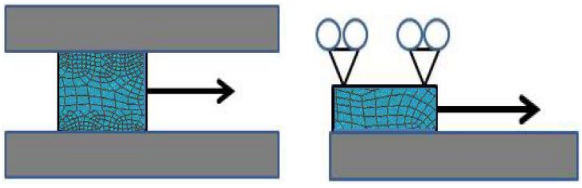


Figure 2: Deformed mesh and conditions imposed in the geometry model

ii. Element type-Plane 223: The element that was used in the ANSYS model was the Plane 223 element. This element has eight nodes with up to four degrees of freedom per node. For the model used in this project only three degrees of freedom were utilized, displacement in the X-direction, displacement in the Y-direction, and a temperature value. This allowed for a coupled structural and thermal solver to be used in ANSYS. This solver takes into account the change in temperature of the material, accounting for the thermal expansion that occurs adding to the normal force generating the frictional heat.

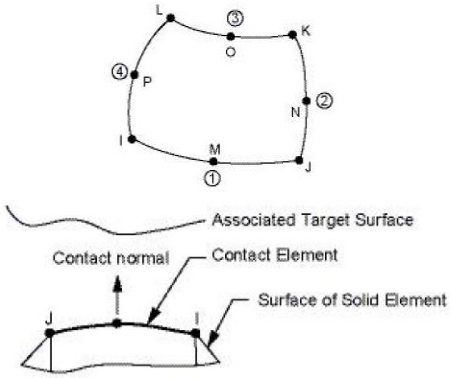


Figure 3: Element shape with surface

iii. Element attributes-Contact 72: In order to properly account for effects of friction, the element Contact 172 is used in ANSYS. This 3-noded element is located on the surfaces of 2-D solid elements with mid-side nodes, as is the case for the element Plane 223 being used for the solid bodies. This element is essential in our analysis because not only does it take into account Coulomb friction, but it is also applicable to 2-D coupled field contact analyses. It is important to notice that this element is nonlinear and thus requires a full Newton iterative solution. This makes FEA calculations expensive, and should be taken into account when considering coupling structural and thermal analyses in which friction is present.

iv. Element segment-Target 169: Frictional effects should only be taken into account when a contact surface touches a target surface, as previously discussed, the contact surface is discretized using element Contact 172. The target surface is thus discretized into target segments using element Target 169. Each target surface is paired with an associated contact surface via a real shared constant set, contact occurs when the contact surface penetrates one of the target segment elements on a specified target surface.

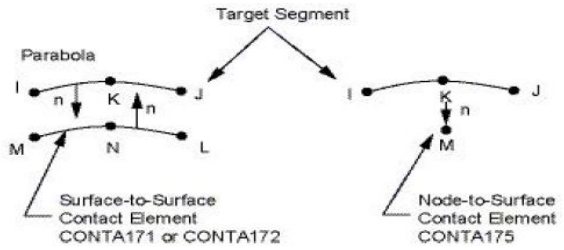


Figure 4:Frictional segment

C. Solution Process

Typically when investigating a thermal stress problem the temperature field induces thermal strains in the structural field, yet the structural strains do not usually affect the temperature distribution. This shows that there is no need to tightly couple the two field solutions. However, when friction is introduced the structural strains directly related to the heat generated at the frictional connection. Due to this double reliance a tightly coupled solver is needed to analyze this model. The method at which ANSYS incorporates the frictional heating is relatively straight forward. The first step ANSYS takes is to use a basic coulomb friction model.

$$\tau = \mu * p + COHE$$

This equation relates the equivalent frictional stress, to the friction coefficient, the contact pressure p, and cohesion sliding resistance COHE. The cohesion sliding resistance is defined as the friction force at zero contact pressure. The ANSYS default for this value was zero, and for this model was maintained at zero due to the approximation that interacting clean metal surfaces have a negligible amount of cohesion. From this equivalent frictional stress the heat generated is calculated.

$$q = (FHTG)\tau V$$

This equation shows the relation to the total heat generated, q, to the fraction of frictional dissipated energy converted into heat FHTG, the equivalent stress, and the sliding rate V. The sliding rate is a value that can be set by the problem being investigated. For this simulation it was set to 0.15m/s. The frictional dissipated energy fraction is defaulted to one by ANSYS, meaning that all the energy generated from this frictional interaction is turned into thermal energy. ANSYS then divides up the heat generated to the two surfaces involved in the friction interaction.

$$q_c = (FWGT)q$$
$$q_t = (1 - FWGT)q$$

Where q_c and q_t represent the heat dissipated to the contact side and heat dissipated to the target side respectively. FWGT is the weight factor for the distribution of the heat between the contact and target surfaces. The ANSYS default is 0.5 representing an even distribution of heat between the two surfaces. This approximation holds for the model in question due to the symmetry of the interaction and the same material for both parts.

In ANSYS utilizing element Plane 223, a tightly coupled solver is used. A tightly coupled solver means that the thermal and structural equations a solved simultaneously at each time step iteration. As compared with a loosely coupled solver which would solve one of the equations first then using results from that run the second equation, then iterate to the next time step. The benefit of using a tightly coupled method allows for a more accurate solution, however the accuracy comes at a price. The stiffness matrix used for the tightly coupled approach is no longer symmetric and introduces a much heavier computational cost. The form of the equation will take a shape as shown.

$$\begin{bmatrix} K_{11}K_{12} \\ K_{21}K_{22} \end{bmatrix}\begin{bmatrix} X_1 \\ X_2 \end{bmatrix} = \begin{Bmatrix} [F_1] \\ [F_2] \end{Bmatrix}$$

In our typical structural or thermal problem the K_{12} and K_{21} are the same which then lends itself to be a symmetric matrix. However in the tightly coupled equation the K_{12} and K_{21} are no longer the same which is the reason for the non-symmetry. However, when investigating a frictional interaction a tightly coupled equation is more ideal since the dependency is cyclical as shown in the figure.5. While investigating the model a special at-tension was paid to the mesh refinement. Quite often, the finite element thermal model requires a finer discretization than the structural model to compute the temperature distribution accurately. Since the ANSYS simulation is a tightly coupled equation and is based all on one mesh, it was important that the model be refined enough to properly account for the thermal and structural effects.

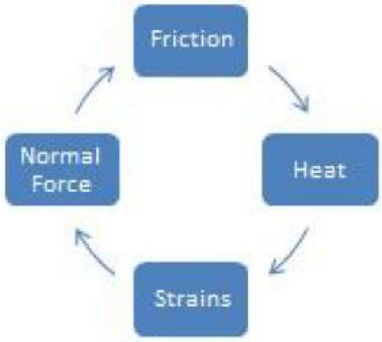


Figure 5:Frictional interaction

The choice of material heavily influences the friction interaction due to the coefficient of friction value and cohesion value. Both of these vary significant from material to material inter actions. For our model consisting of a Steel to Steel interaction the coefficient

of friction is approximately 0.80 and the value for the cohesion zero.

III. RESULTS

A. Verification of mesh convergence

In order to ensure that the solution is converging, the solution must be ran again using a finer mesh. The plot below shows the results, using two refined meshes. The plot shows that the results converge when the element size is halved. The original mesh yields results that are an order of magnitude off the converged values.

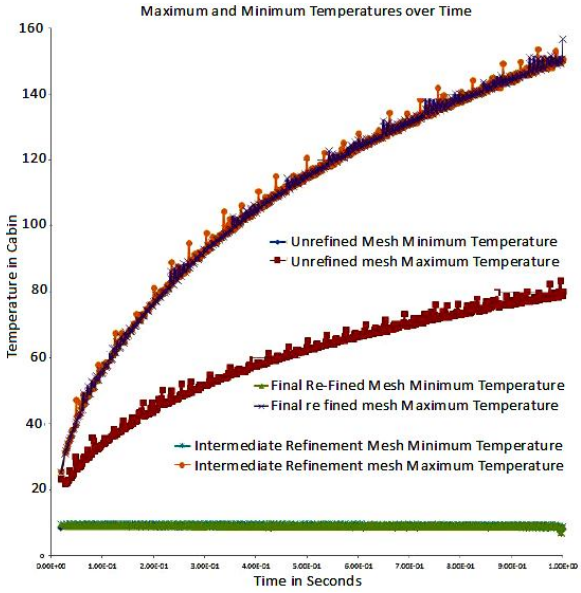


Figure 6:Time vs Thermal conductivity and diffusivity

B. Analytical Solution

To verify the model we applied an analytical solution to solve for the temperature at the frictional interface. The model approximated the frictional input as a constant value based on the force that would be needed to replicate the initial displacement set in the ANSYS simulation. For further simplification purposes, we assumed the sliding piece to be a semi-infinite solid, given these two conditions, we can approximate the analytical solution from Carslaw and Jaegar (1959, p. 75) specifically for the contact point on the block:

$$T(t) = T_i + \frac{2q_o}{k}\left(\frac{\alpha t}{\pi}\right)^{\frac{1}{2}}$$

Where T_i is the initial temperature, t is the time, k is the thermal conductivity, and α is the thermal diffusivity. Figure.6 shows the resulting graph. The overall shape of the graph agrees with our ANSYS solution. However, the values do not. This is likely due to our approximation of a semi-infinite solid, which the sliding block is clearly not. However, we are confident this authenticates our ANSYS model due to the fact that the analytical solution is of the same order of magnitude and shape as our ANSYS solution.

C. Verification of Matlab vs. Finite Element Analysis

In verifying our results from ANSYS, we developed a simplified model that was implemented in MatLab. We approximated the heat flux due to friction to be constant, as was done in the analytical solution. Since no other boundary conditions were taken into consideration, the sliding block could be modeled as a one-dimensional boundary value problem. As a result, the MatLab compilation 1DBVP was modified appropriately for the problem.

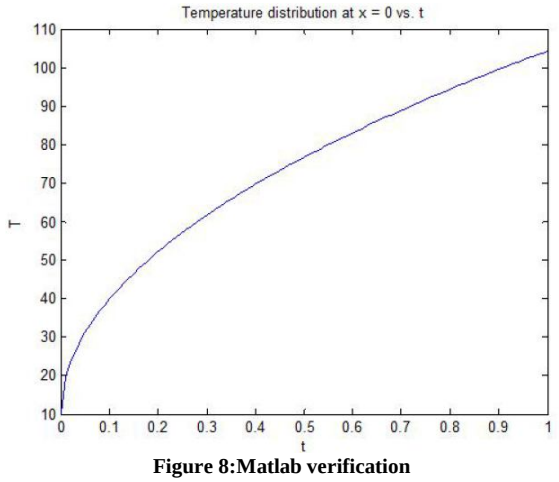


Figure 8:Matlab verification

The elements chosen were 1D axial element that runs vertically. Heat flux due to friction was added as a natural boundary condition on the bottommost element and the initial temperature was set to 10^0. A numerical time integration scheme was implemented as shown on slides 45^0 to 50^0, utilizing the Euler method, setting θ=0 and using a lumped mass matrix.

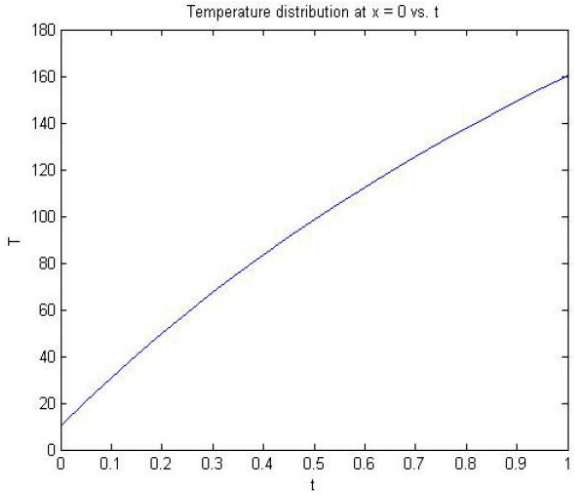


Figure 9 :Ansys verification

We compared the results at the contact boundary node to the maximum of the ANSYS results as these also occurred at a boundary node.

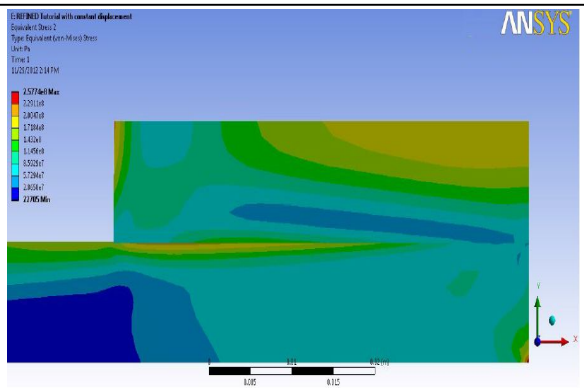


Figure 10:With thermal effect

As can be seen by comparing the plot below the previous one, they agree quite well, fortifying our ANSYS model.

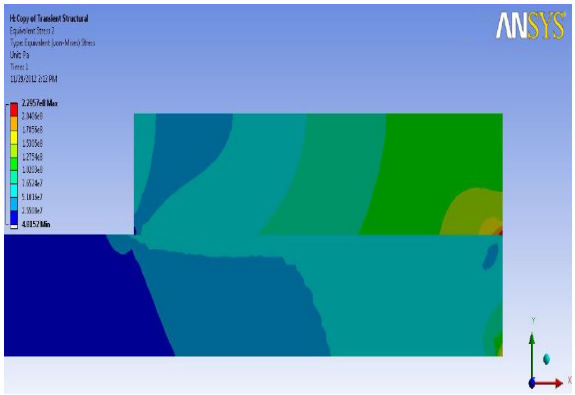


Figure 11:Without thermal effect

CONCLUSION

The current study has demonstrated two different approaches for examining the results from the ANSYS simulation we see the inclusion of thermal frictional effects drastically affect the results. In comparing the results we used the top edge and took only the central values. We found that on the non-frictional model the average stress found was approximately 81 MPA. In the frictional model the stress was approximately 150 MPA. This was roughly a 46.

REFERENCES

[1] S. Abbasi, T. Shahab, V. Tore and S. Ulf, "Temperature and thermoelastic instability at tread braking using cast iron friction material", Wear, 314 (2014), pp.171-180.

[2] D. B. Mozor, Y. H. Hatzor, S. D. Glaser and J. S. Carlos, "Thermally vs. seismically induced block displacements in Masada rock slopes", International Journal of Rock Mechanics & Mining Sciences, 61 (2013), pp. 196–211

[3] A.Molinari, R. Cheriguene and H. Miguelez, "Contact variables and thermal effects at the tool-chip interface in orthogonal cutting", International journal of Solids and Structures, 49 (2012), pp. 3774-3796.

[4] J. Liu, L. L. Ke, and Y. S. Wang, "Two-dimensional thermoelastic contact problem of functionally graded materials involving frictional heating", International Journal of Solid and Structures, 48 (2011), pp. 2536-2548.

[5] Eriki Anand. K, R. Ravichandra and Edilan. M, "Spur Gear Crack Propagation Path Analysis using Finite Element Method:, IMECS, vol.2, pp. 1362-1367.

[6] Coupled-Field Analysis Guide: ANSYS Inc. Nov.2009.

[7] Contact Technology Guide: ANSYS Inc. Nov. 2009

[8] R. Cagliero, A. Santoro and G. Maizza, Mechanical Behavior of TiAl Spherical particles Including Friction Effect, Dipartimento di Scienza dei Matereriali ed. Ingegeneria Chimica, Politecnico di Torino, 2010.

[9] D. Pramote and Earl A. T., Improved Finite Element Methodology for Integrated Thermal Structural Analysis, Nasa Contractor Report 3635, Nov. 1982.

★★★