A Simple Weak-Field Coupling Benchmark Test of the Electromagnetic-Thermal-Structural Solution Capabilities of LS-DYNA® Using Parallel Current Wires

William Lawson and Anthony Johnson
General Atomics Electromagnetics

Abstract

To begin learning the coupled field capability of LS-DYNA and validate results, a simple simulation of parallel wires carrying current was run. The magnitude of the current in the wires is such that the coupling between the electromagnetic (EM), thermal and structural fields is weak, in the sense that the coupling is taken to be one way. That is, there is no feedback amongst the three field solutions. This allows us to compare LS-DYNA code and known analytical results for code validation to build confidence that the code is being correctly used. LS-DYNA results are also compared to ANSYS results when no analytical results are valid. In addition, this simulation allowed us to test the transfer of EM generated Ohmic heating to the thermal field, and the transfer of EM generated forces to the structural field, a necessary process for coupling fields. Furthermore, to be able to compare the code and analytical results, temperature-dependent material properties have not been included – a decent approximation with the low currents used. The set-up of the coupled field model is discussed. Comparison of the LS-DYNA code and analytical results show good agreement where applicable. Comparison with ANSYS results is also good.

Introduction

With EM-thermal-structural coupled field capability, pulsed current simulations can be run entirely within LS-DYNA by applying a single current versus time load. Field coupling is achieved by first transferring the EM Ohmic loads to the thermal solver as thermal input loads. Then the calculated thermal loads, along with the already calculated EM Lorentz forces, are transferred over to the structural solution as structural input loads, thereby completing the 3-field coupling (1-way) process. Figure 1 illustrates the load coupling transfer process. Solid lines in the figure represent the load transfer used in this paper. The dashed lines represent capability (full 2-way feedback) that exists in LS-DYNA but is not tested in this paper.

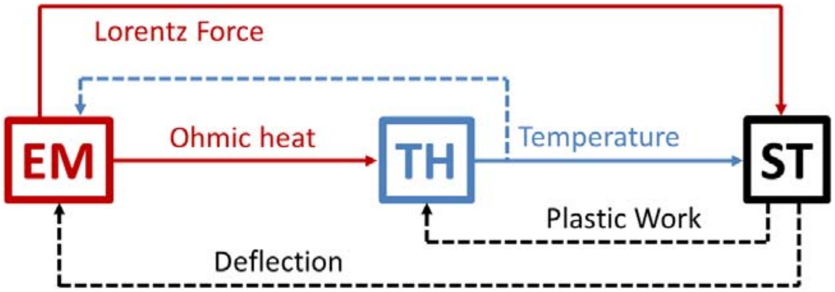


Figure 1. Result Transfer between Electromagnetic, Thermal and Structural Solvers. The solid lines represent result transfer tested in this paper. The dashed lines represent capabilities that exist in LS-DYNA but are not tested by the problem simulated in this paper.

To learn the setup of an EM-thermal-structural simulation, and to verify that it was done correctly, two LS-DYNA simulations of parallel current carrying wires were run. This scenario is pictured in Figure 2. Two versions of this analysis were conducted with different boundary conditions. The first simulation applied current through finite length parallel wires. The second simulation approximated infinite length parallel wires and applied the same current through both wires. The details of the differences between these two simulations will be described later. Results of both simulations were compared to analytic calculations, which are presented in the next section. Where an exact analytic calculation did not exist, the LS-DYNA simulations were compared to the same problem solved in ANSYS.

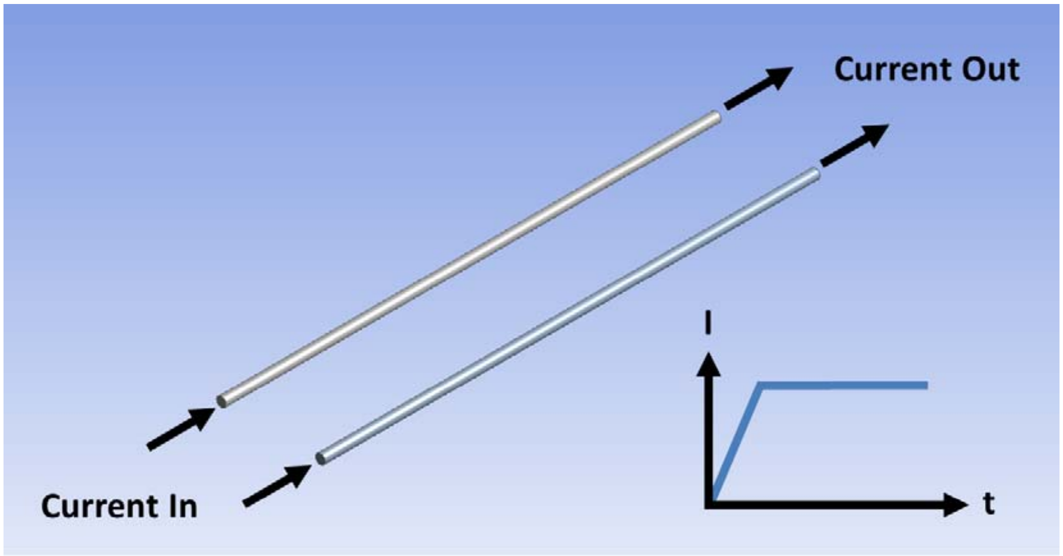


Figure 2. Parallel Wires Carrying Current and Current Profile Versus Time

It is important to note that the reason for running a simulation of parallel, current carrying wires is that many of the results can be compared to basic analytic calculations for all three fields (electromagnetic, thermal and structural). This allows for one simulation to benchmark all three fields and their associated coupling. Verifying the code-generated results of a simple model against analytical results is always wise as a means to build confidence in both the use of the code and in the code results, before moving onto something more complicated, since in general no such analytical results exists for comparison.

Only a few keyword cards are required to set up the LS-DYNA EM simulation.

- *EM_CONTROL was used to enable the eddy current solver and set the number of cycles between updates of the EM-FEM and EM-BEM matrices. Both of these were set such that the matrices are only calculated at the beginning of the simulation since displacements were expected to be small and EM material properties did not include temperature dependence.
- *EM_CONTROL_TIMESTEP was used to select automatic time step calculation
- Two *EM_CIRCUIT cards, one for each wire, tell the solver where the current goes in, where it goes out, and how it varies with time
- *EM_MAT_001 was used to define each wire as a conductor and define the electrical conductivity. The properties for wire 1 and wire 2 are the same, but they need to be defined as two different materials for the EM solver

The setup of the thermal simulation also requires only a few keyword cards.

- *CONTROL_SOLUTION was used to select a coupled thermal-structural analysis
- *CONTROL_THERMAL_SOLVER was used to select the solver type and convergence tolerance
- *CONTROL_THERMAL_TIMESTEP was used to select a constant time step size
- *INITIAL_TEMPERATURE_SET was used to initialize both wires to room temperature
- *MAT_THERMAL_ISOTROPIC was used to set the thermal conductivity and specific heat capacity of the wires

The setup of the structural solution includes keyword cards that will be familiar to most LS-DYNA users. These cards won't be discussed here for that reason, and since the keyword deck is included at the end of this paper.

Analytic Calculations

For the case of two parallel wires carrying a DC current, analytic solutions can be written down for all three fields as well as the net effect of the coupling between the fields. This section will summarize these analytical equations.

In the EM-field solution, element force densities are calculated and converted to nodal forces applied in the structural field. The electromagnetic Lorentz force per unit length w between the wires (infinite in length) is given by the equation below, where I is the current and d is the distance between the wire centers. It is assumed that the distance between the wire centers is much greater than the radii of the wires so that the finite wire thickness can be neglected. Under this assumption we obtain for w

$$w = \frac{\mu_0 \, I^2}{2 \, \pi \, d}$$

In this paper, plots of force will include a minimum, nominal and maximum analytical calculation. Since the model presented here uses finite radii wire, the force between the wires can't be exactly calculated. The nominal force prediction uses the center-to-center distance between the wires in the force calculation. The minimum and maximum force predictions assume the distance is between the outer edges of the wires and the inner edges of the wires, respectively, as shown in Figure 3.

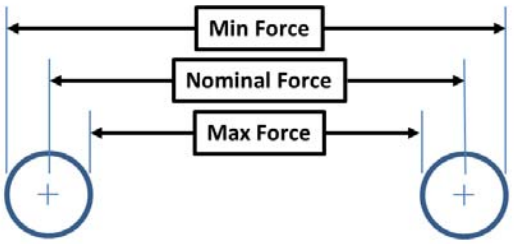


Figure 3. Distances Used for Minimum, Nominal and Maximum Force Between Wires Analytical Calculation

Additionally in the EM-field solution, Ohmic heating is calculated and later applied in the thermal field solution as an input. The Ohmic heat rate *Power* on a wire is given below, where *I* is the current, ρ_e is the material electrical resistivity, L is the wire length and *A* is the wire cross sectional area.

$$Power = I^2 R = I^2 \frac{\rho_e L}{A}$$

In the thermal-field solution, the temperature increase in each wire is determined from the following equation, where *ΔT* is the temperature change, m is the wire mass, *c* is the material specific heat capacity (assumed temperature independent), *Power* is defined in the preceding equation and *t* is time.

$$\Delta T = \frac{1}{m\,c} \int_o^t Power\,dt$$

The structural response of the two wires can be treated as Bernoulli-Euler beams with a uniform load provided by the electromagnetic field. The maximum static deflection d_{max} of a wire fixed at both ends is given by the equation below, where *w* is the uniform force per unit length on the wire calculated above, L is the length of the wire, *E* is Young's modulus and *I* is the wire area moment of inertia.

$$d_{max} = \frac{w\,L^4}{384\,E\,I}$$

Finite Length Wires Simulation

The first benchmark simulation runs current through finite length parallel wires. The current orientation is parallel, and as a result attractive forces are generated between the wires. The simulation results are in good agreement with the analytic calculations for Ohmic heating and temperature rise as shown in Figure 10 and Figure 11, respectively. Since the wires are finite in length, end effects are expected and evident as shown in Figure 9, and as a result the Lorentz force and structural deflection do not match the analytic calculations.

Because the wires used in the LS-DYNA simulation are finite in length and have non-zero thickness, it is not surprising that the Lorentz force agreement between the code results and the analytical (infinite wire lengths and zero wire thickness) calculation is not great. Thus to avoid the inherent limitations cited, we have also compared the LS-DYNA results with those from an ANSYS simulation of the exact same problem. This simulation provides an independent calculation of the Lorentz force. The ANSYS simulation also provides an independent calculation of the heat generation rate and temperature increase, although the LS-DYNA results are expected to match these analytic calculations.

To make the comparison between the LS-DYNA and ANSYS simulations as apples-to-apples as possible, identical wire meshes were used. ANSYS Workbench was used to create the ANSYS mesh of the two wires and a large volume of air surrounding the wires. The LS-DYNA mesh is created from the Workbench mesh by unselecting the air and converting the wire nodes and elements to the LS-DYNA keyword format with ANSYS Parametric Design Language code within ANSYS. The LS-DYNA solver automatically models the air mesh with boundary elements. The geometry and mesh including the air are shown in Figure 4 and Figure 5, respectively. Twelve hexahedron elements are used to model each wire diameter as depicted in Figure 6. The length of the wire is modeled with 100 hexahedron elements.

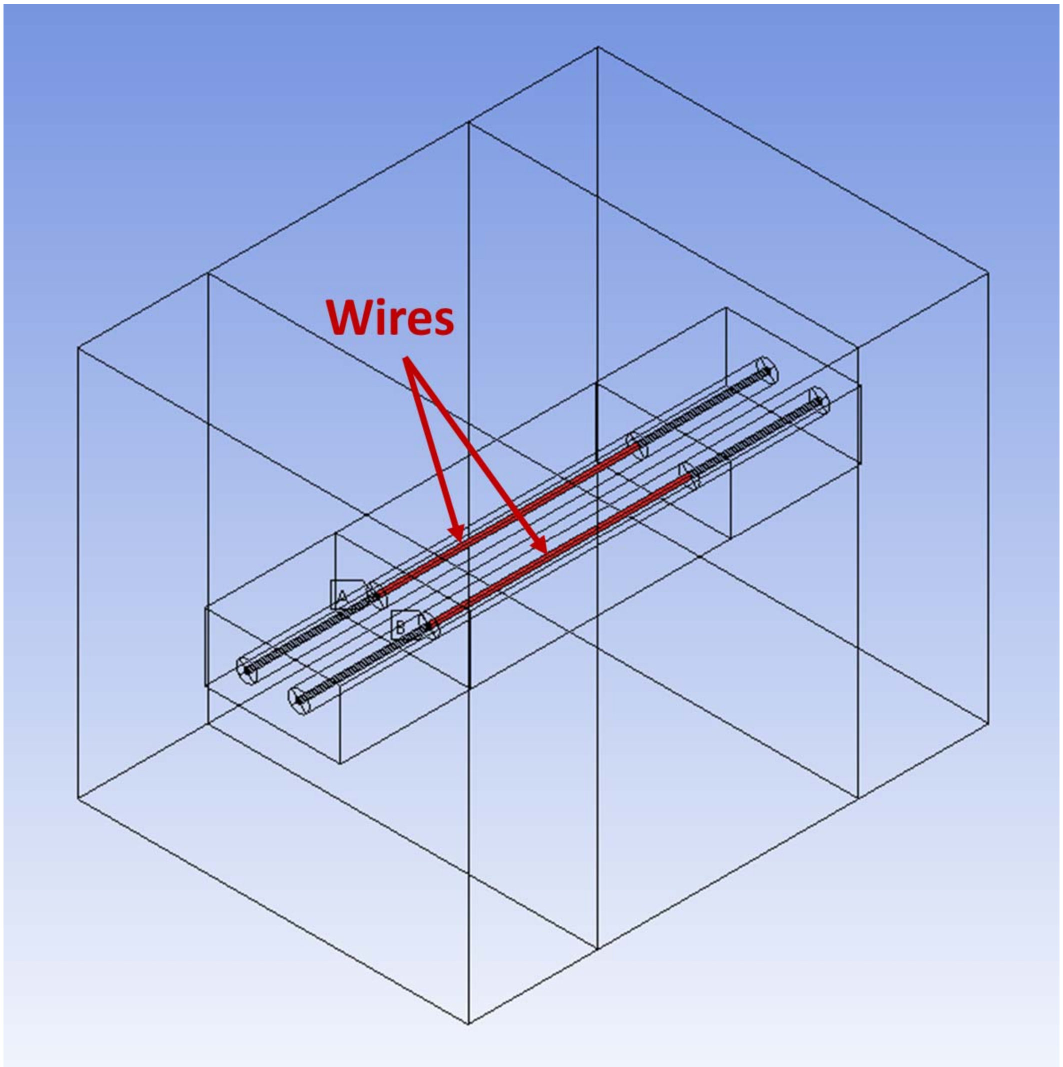


Figure 4. Finite Length Model Geometry Including Air

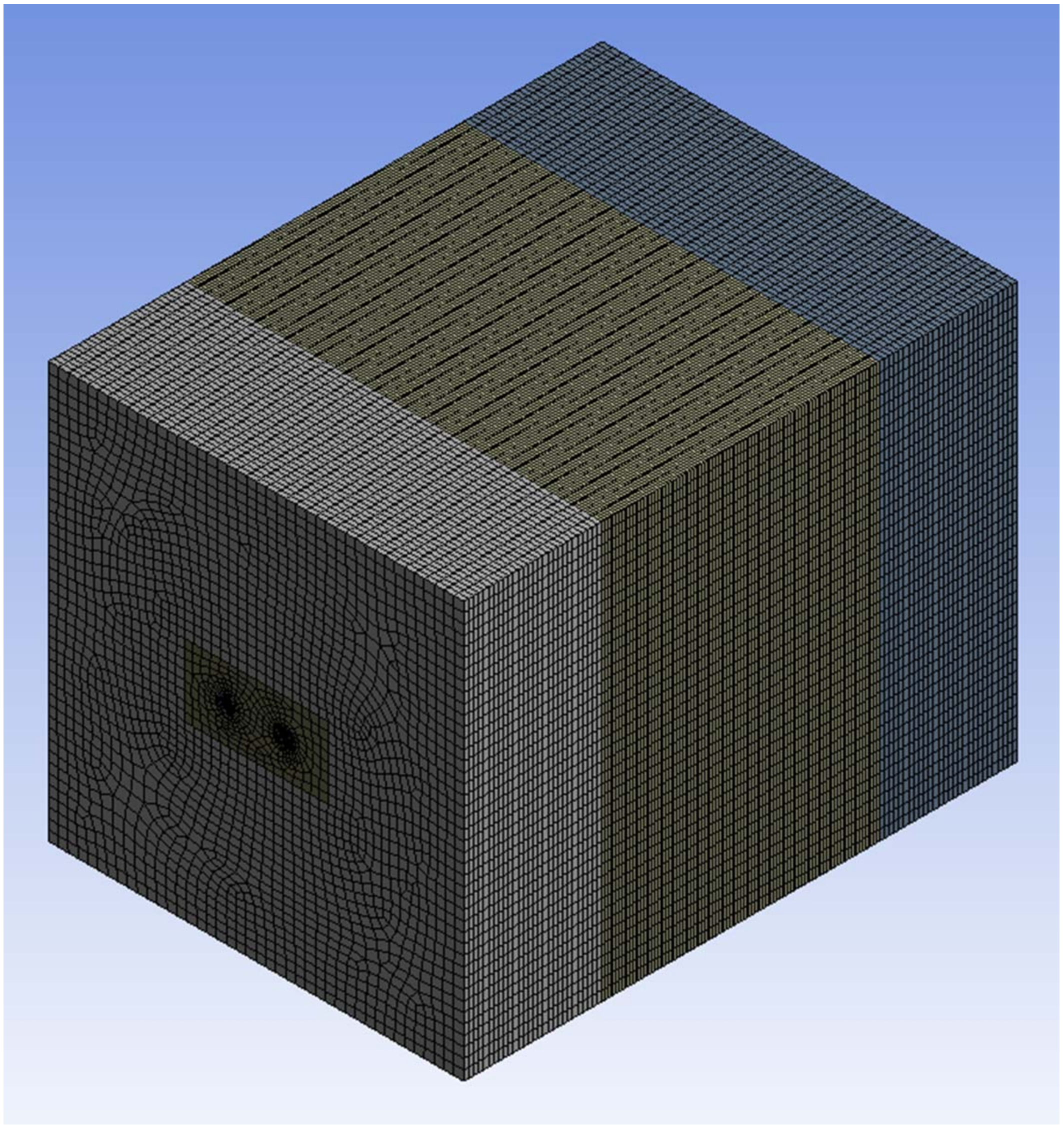


Figure 5. Finite Length Model Entire Mesh Including Air

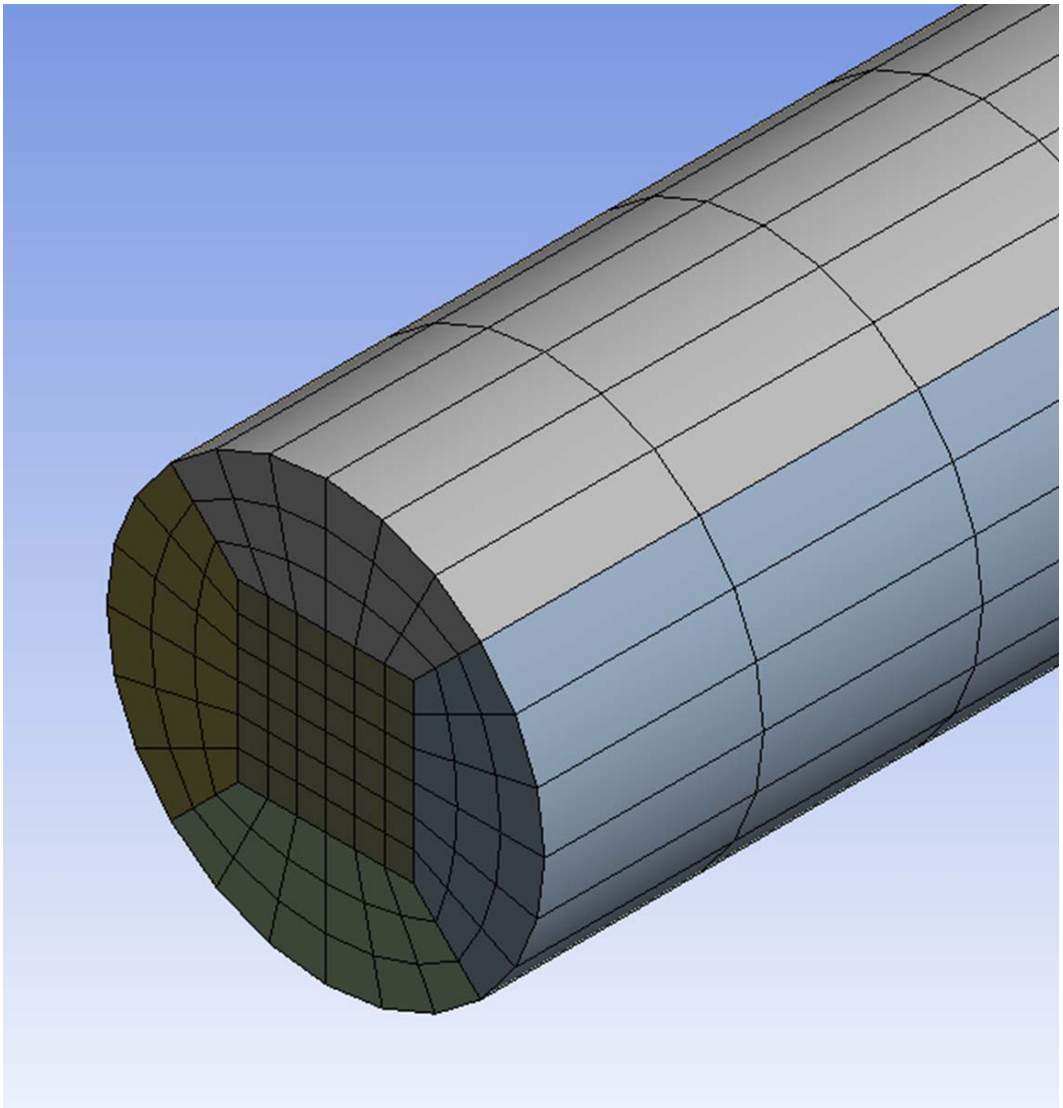


Figure 6. Finite Length Model Detail of Wire Mesh

The ANSYS and LS-DYNA models are shown in Figure 7. The most obvious difference between the two models is the large box of air surrounding the wires in the ANSYS model. The ANSYS model also includes EM boundary conditions on the box exterior faces, but the air volume is intended to be large enough that the air volume looks approximately infinite so that the magnetic field runs tangent to the air walls. As expected, the ANSYS model has more elements as a result of the air enclosure, but this does not necessarily equate to longer EM solution times, since the LS-DYNA air boundary element matrix is denser than the ANSYS air finite element matrix. The advantage of the boundary element method used in LS-DYNA comes in easier meshing and the ability to simulate large structural deflections.

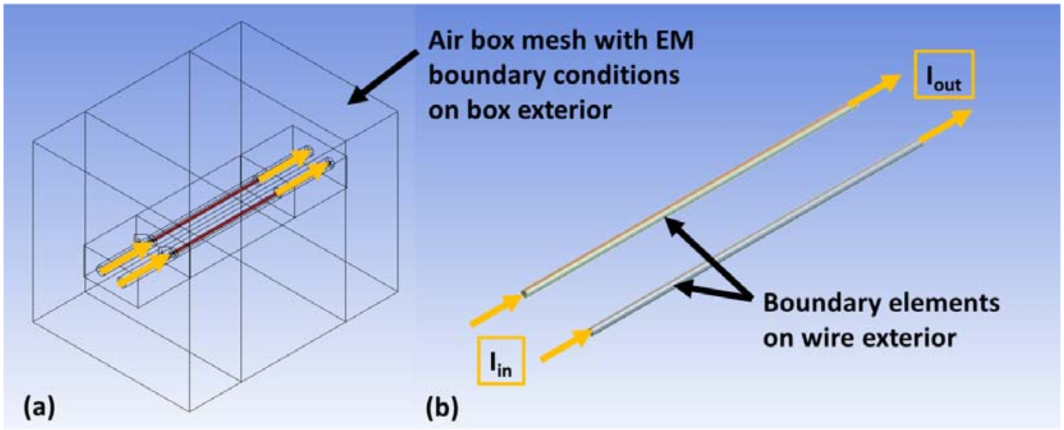


Figure 7. ANSYS (a) and LS-DYNA (b) finite length wires models

Listed in Table 1 are the geometry, material and load inputs for both the LS-DYNA and ANSYS models.

Geometry	
Wire Radius (m)	0.005
Wire Length (m)	0.5
Center-to-center Distance Between Wires (m)	0.1
Material	
Wire Young's Modulus (N/m^2)	1.250E+11
Wire Density (kg/m^3)	8900
Wire Specific Heat Capacity (J/kg K)	385
Wire Thermal Conductivity (W/m-K)	390
Wire Electrical Resistivity (Ohm-m)	2.00E-08
Wire Electrical Conductivity (S/m)	5.00E+07
Load	
Max Current (A)	20000
Current Rise Time (s)	0.01

Table 1. Finite Length Wires Simulation Inputs

Since the current profile includes a rise time, it is important to consider the diffusion skin depth and ensure that the mesh has at least 2-3 elements per skin depth to accurately model the current diffusion at early times. When current is applied quickly, it initially flows within 1 skin depth of the surface and gradually diffuses through the thickness of the conductor with increasing time. The equation for calculating the skin depth is given below. Observe that the skin depth (δ) decreases, thus requiring a smaller element thickness to capture the current behavior correctly, as the current frequency (f) increases (rise time decreases) and as the electrical conductivity (σ) of the material increases. μ is the permeability of the wires and is taken to be that of free space and remains constant.

$$\delta = \frac{1}{\sqrt{\pi f \mu \sigma}}$$

Taking the 0.01 second rise time to be ¼ period, the input frequency for this simulation is 25 Hz. The resulting skin depth is 0.0142 m, which is nearly 3X the wire radius. Diffusion effects are not important for this EM simulation. The through thickness element dimension will accurately capture the diffusion at early times and the DC approximation in the analytical results is applicable.

The time history of the attractive Lorentz force per unit length between the finite length wires will not match the analytic calculation for infinite wires for the reasons cited previously which is confirmed in Figure 8. Observe the good overall agreement between the LS-DYNA and ANSYS force per unit length time histories. On the flat top, where the largest disagreement occurs, the difference between the LS-DYNA and ANSYS Lorentz force is less than 2%. Also observe that both the LS-DYNA and ANSYS results calculate a lower attractive force between the wires than the analytical calculation. Why this is the case will become clear shortly.

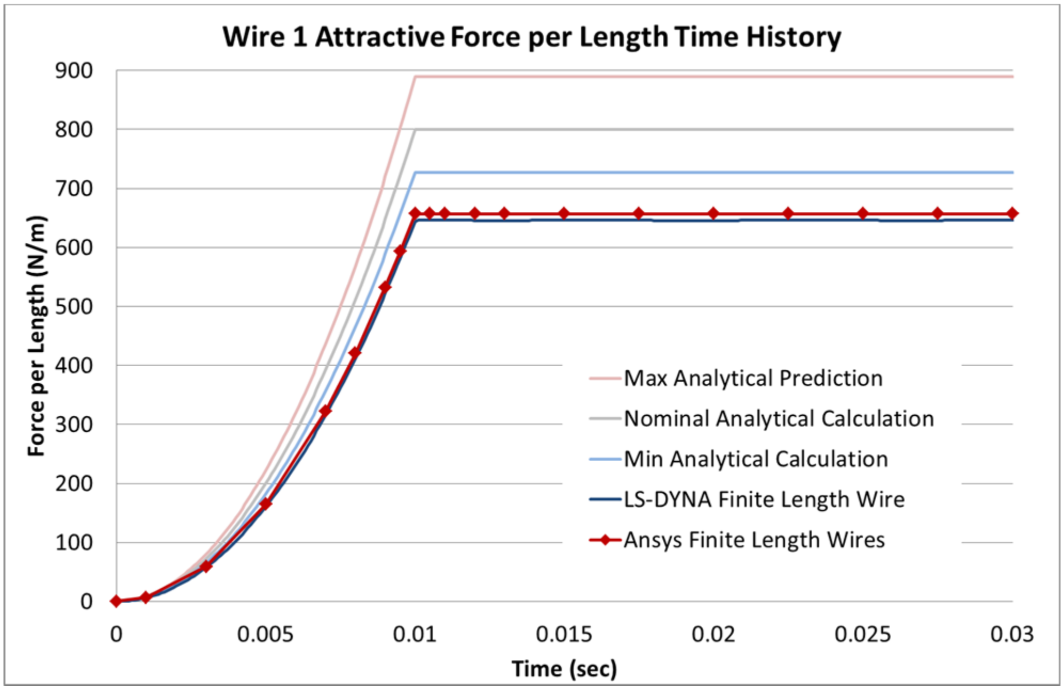


Figure 8. Time History of Attractive Force per Unit Length between Finite Length Wires

While Figure 8 shows a good match in total force between the LS-DYNA and ANSYS simulations, it does not present information on the distribution of the force. Figure 9 shows the force per unit length along the length of the wire at maximum current. Since the 3-dimensional wire mesh is simply an extrusion of a 2-dimensional mesh, the force in each axial section of elements can be summed up and then divided by the length of the section to calculate force per unit length. Each triangle or diamond marker in Figure 9 represents one of the 100 element sections along the axial length of the wire. Aside from the five element sections at each end of the wire, agreement is very good between the LS-DYNA and ANSYS simulations. The source of the difference at the ends of the wire is unknown. Figure 9 makes it clear that end effects were a primary contributor to the lower total forces per unit length for the code results (as compared to the analytical calculations) depicted in Figure 8. It is also clear that the 2% difference in total force between the LS-DYNA and ANSYS simulations is mostly due to differences in force at the wire ends. Agreement away from the ends is much better than 2%.

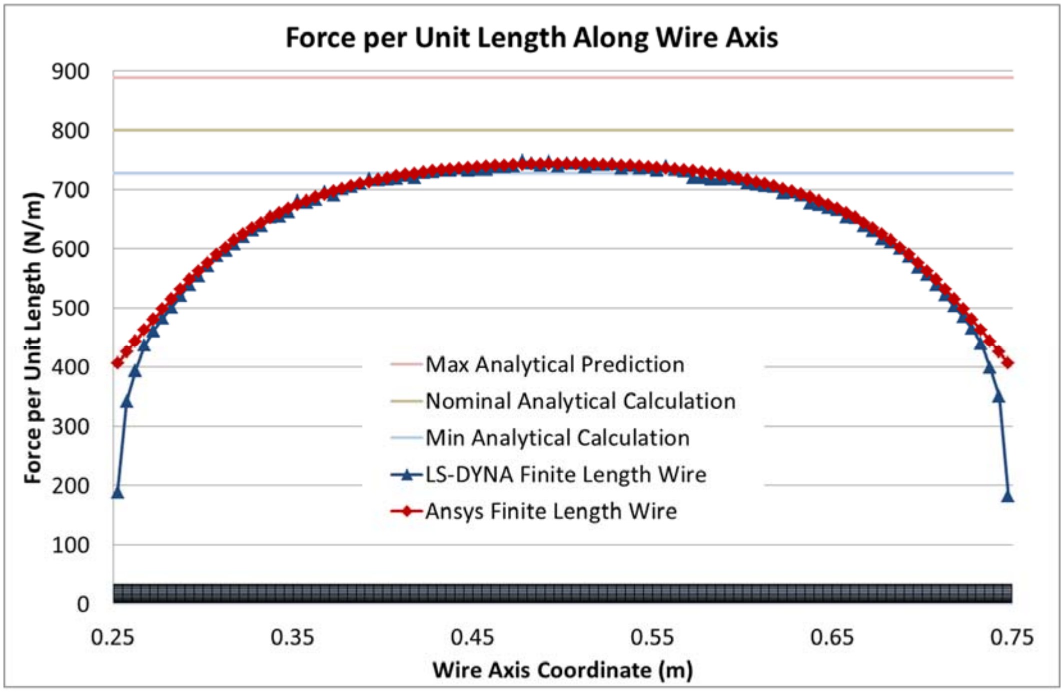


Figure 9. Attractive Force per Unit Length between Finite Length Wires along Wire Length at Maximum Current

The total wire heat generation rate in the LS-DYNA simulation was calculated by summing the heat generation rate of all the elements in each wire. The LS-DYNA simulation under-predicts the analytical heat generation rate at maximum current by 2.1%. The analytical calculation used the actual cross sectional area of the finite element mesh. A finite element mesh can't perfectly represent a circle, and as a result has a slightly lower area than a perfect circle – 1.1% lower for the mesh used in this simulation. This lower cross sectional area leads to a greater wire resistance, which leads to a greater heat generation rate as compared to a perfect circular cross section wire. The ANSYS simulation result (not shown in the plot) is a near perfect match with the analytical calculation, differing by less than 0.01% at maximum current.

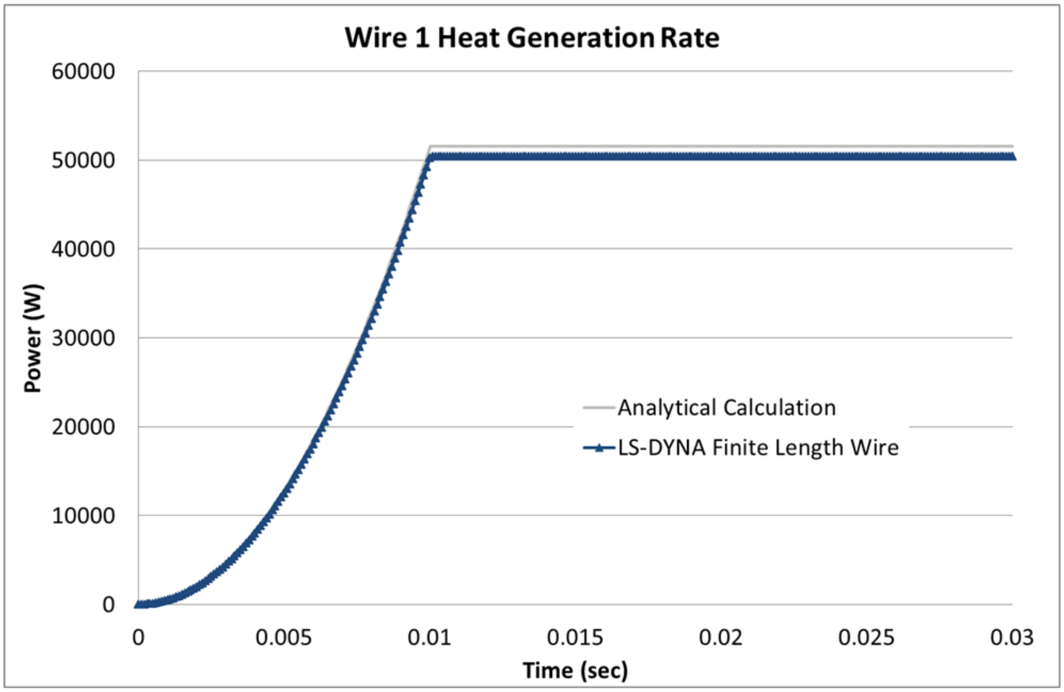


Figure 10. Time History of Heat Generation Rate in One Finite Length Wire

The average wire temperature was calculated from the LS-DYNA simulation by summing all the nodal temperatures in each wire and then dividing by the number of nodes. This is a reasonable average temperature approximation for this model given the relatively uniform mesh and relatively uniform temperature distribution. This method would not be as accurate for models with highly non-uniform meshes, localized temperatures and/or diffusion.

The wire average temperature rise at the end of the simulation under-predicts the analytic calculated temperature rise by 2.3%, which is nearly consistent with the simulation under-predicting the heat generation rate by 2.1%. The under-prediction in temperature rise and heat generation rate should be identical, and the minor difference is likely due to the nodal temperature averaging technique used to calculate the average temperature. Like the heat generation rate analytic calculation, the average temperature calculation uses the actual geometry from the finite element model. The modeled cross sectional area and mass are both 1.1% lower than the ideal geometry, and reducing these 2 numbers increases the expected temperature change.

The ANSYS average temperature was calculated by taking the volume averaged temperature over each wire. The ANSYS temperature time history is not shown in Figure 11 since it lies nearly on top of the analytical calculation time history. The ANSYS simulation temperature rise is 0.2% greater than the analytic calculation.

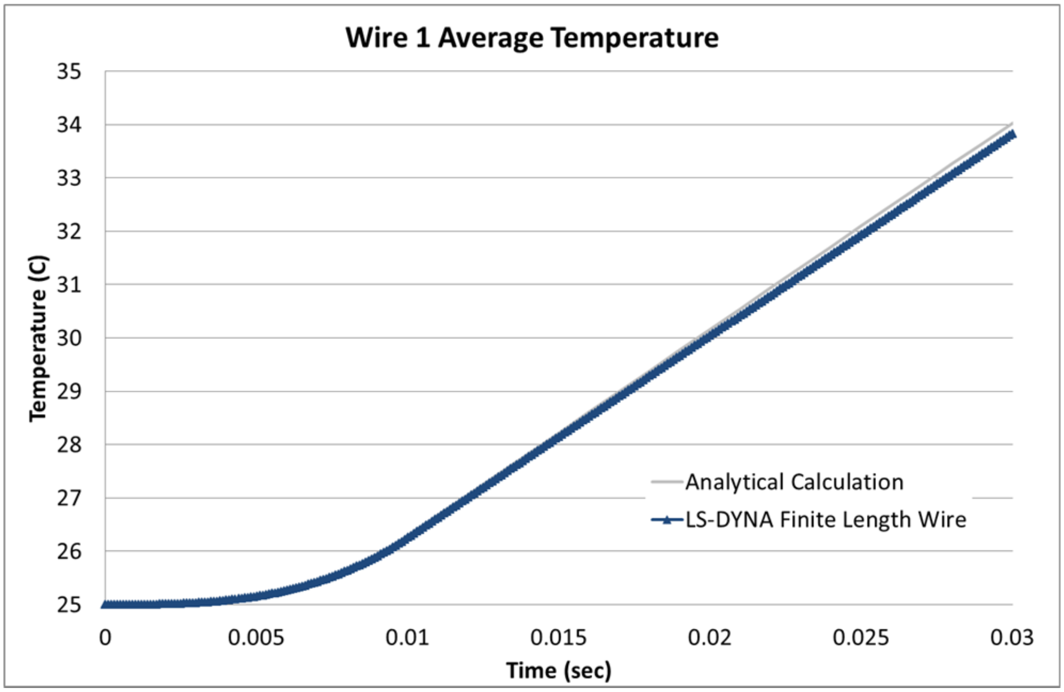


Figure 11. Time History of Finite Length Wire Average Temperature

The results of the LS-DYNA coupled field simulation are in good agreement with analytic calculations and the ANSYS coupled field simulation. The LS-DYNA calculated total attractive force between the wires is 2% lower than the force calculated with the ANSYS coupled field simulation. The LS-DYNA and ANSYS force distribution along the length of the wire is a very good match away from the wire ends. At the wire ends, the LS-DYNA calculated forces are lower than the ANSYS calculated forces. There is no reason to believe that either simulation is more accurate than the other. The fact the total force, and force distribution along the length of the wires, is very similar between the two codes is encouraging.

The LS-DYNA calculated heat generation rate and temperature rise are 2% lower than the analytic calculated values. One possible source of this difference is the convergence tolerances used in the implicit EM and thermal solvers. However, from the perspective of designing and simulating a pulsed current device, the difference between the LS-DYNA code generated results and the analytic calculations is small and more than acceptable.

Deflection results have not been presented for the finite length wires simulation since there is no basic analytic calculation for the non-uniform applied load. Deflection results will be presented in the next section for the semi-infinite wires simulation, as they are necessary for verification that the Lorentz forces are properly applied in the structural solution. Stress has also been omitted since there is no basic analytic calculation for the non-uniform loading applied to the finite length wires, and since a high number of through thickness elements are required to accurately predict the bending stress in the wires.

Semi-Infinite Wires Simulation

As stated earlier in this paper, the analytic calculation for force per unit length between parallel wires carrying current neglects end effects, or in other words, assumes very long wires. End effects were clearly significant for the finite length wires simulation presented in the previous section as seen in Figure 9. In fact, the length over which the end effects act can be approximated from Figure 9. Due to the expected end effects, a valid comparison between the LS-DYNA Lorentz force and the analytically calculated Lorentz force could not be made. Also, because of the non-uniform Lorentz force distribution along the length of the wire, the structural deflection of the wire mid-point could not be checked using the textbook formula for a uniformly loaded beam. The LS-DYNA and ANSYS simulations presented in this section model semi-infinite length wires so the code generated forces and deflections can be compared to the analytical solutions.

The LS-DYNA semi-infinite length wires model was created by adding 0.75 meter rigid wires to each end of the 0.5 meter elastic finite length wires model as shown in Figure 12. These rigid wires were structurally constrained in all degrees of freedom. The long rigid wires were used to accomplish a few things. First, they are long enough to keep the end effects away from the elastic center section, resulting in the uniform load on the elastic section that is needed to validate Lorentz force and the wire mid-point deflection. The rigid wires also constrain the ends of the elastic section, which is necessary to provide the fixed boundary condition assumed by the wire mid-point deflection calculation. Finally, the rigid wires were used to minimize the structural solution CPU time. Both the center and the ends of the wires were modeled using identical thermal and electromagnetic material models.

In ANSYS, a semi-infinite length model was created with two 0.5 meter long wires surrounded by air. In this model the air ends where the wires end in the wire axis direction, and EM boundary conditions were used to simulate the wires as infinite length. This was accomplished by enforcing flux parallel boundary conditions on all of the outer boundaries of the air volume.

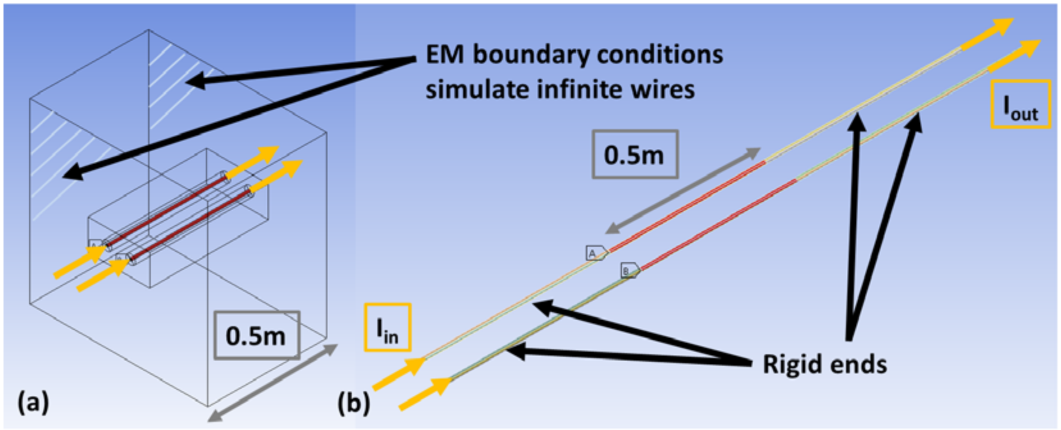


Figure 12. Semi-Infinite Wires Model Geometry for Ansys (a), and LS-DYNA (b)

The force per unit length calculated by the LS-DYNA simulation is 0.7% greater than the force per unit length calculated in the ANSYS simulation (averaged over the length of the wire). This comparison was made at the end of the simulation, with the maximum current applied to the wires. The LS-DYNA simulation also has a slight (±1.2%) variation in force over the length of the wire. Both simulations show forces per unit length within 1% of the nominal analytic calculation, which used the wire center-to-center distance for the force calculation.

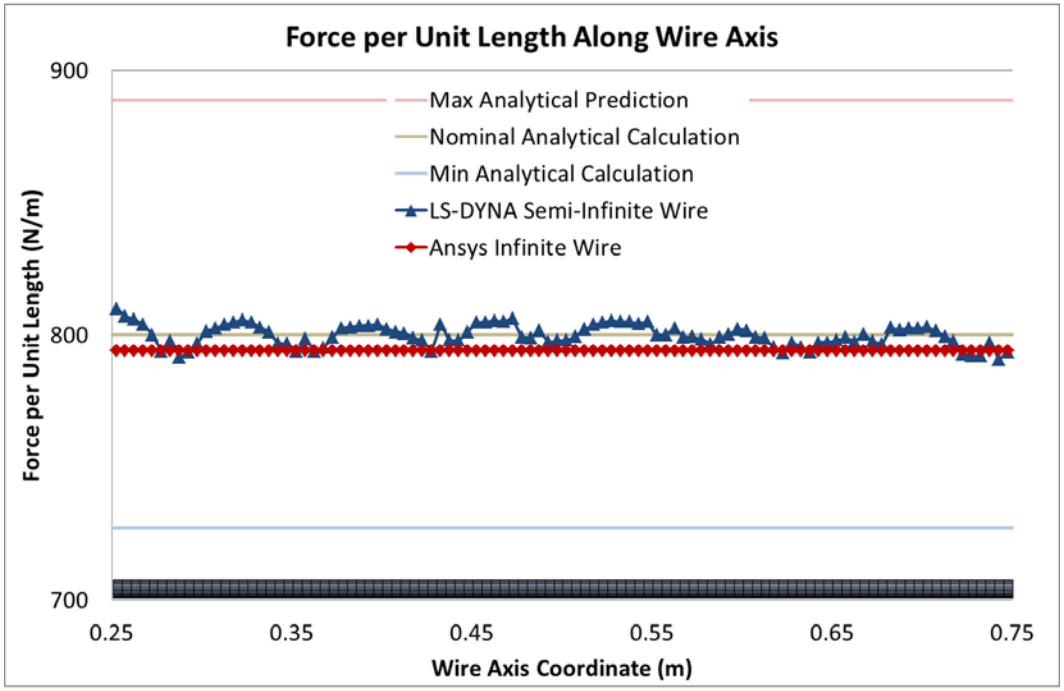


Figure 13. Attractive Force per Unit Length between Semi-Infinite Wires along Wire Length at Maximum Current

The analytically calculated wire midpoint deflection shown in Figure 14 assumes zero dynamic amplification, so the LS-DYNA and ANSYS calculated deflections are expected to oscillate about the analytic calculated time history. The LS-DYNA wire midpoint deflection at full current is oscillating about a value 3% above the analytic calculated deflection, which is 3% more than expected since the LS-DYNA force per length is within 0.1% of the analytic calculated nominal value. Interestingly, the natural frequency measured in the LS-DYNA simulation is about 3% higher than the analytically calculated value. So, the wire has a higher natural frequency (implying high stiffness) and higher deflection (implying low stiffness) than calculated analytically, which is not expected. Sorting out these minor differences likely calls for a mesh sensitivity study. The ANSYS simulation uses 20 implicit structural load steps (each diamond marker in the ANSYS wire midpoint deflection time history curve is a load step). The ANSYS calculated midpoint deflection is oscillating near the analytically calculated midpoint deflection, but the time stepping is too coarse to accurately simulate the structural response.

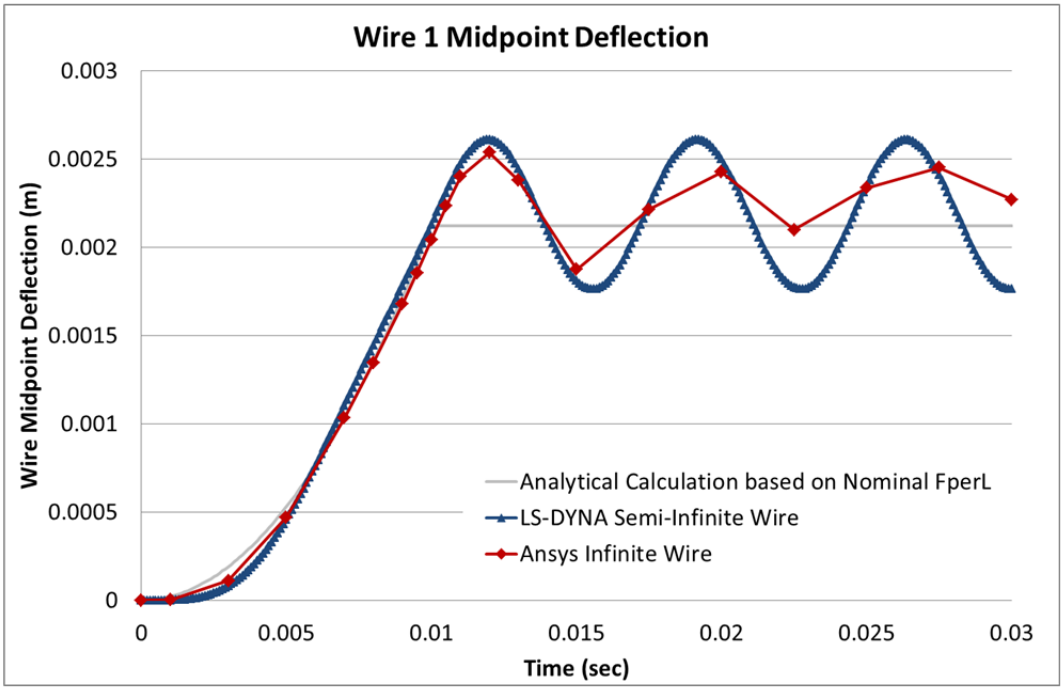


Figure 14. Semi-Infinite Wire Midpoint Deflection Time History

The force results of the LS-DYNA semi-infinite wires simulation compare well with the ANSYS and analytical calculations. The wire midpoint deflection is 3% higher than expected, but close enough to conclude that the Lorentz forces are being properly applied. Combined with the finite length wires simulation, all LS-DYNA results are within 2-3% of the ANSYS and analytical results (where applicable). Certainly additional simulations could be run to understand the minor differences between the LS-DYNA, ANSYS and analytical calculations. One simulation on that list would better approximate the analytical calculation zero thickness wire assumption and reduce the wire radii relative to the center-to-center distance between the wires. Another set of simulations could examine the sensitivity of the results to mesh density. However, a 2-3% difference between the LS-DYNA, ANSYS and analytical calculations is “close enough” for most applications to proceed with simulating more complicated devices in the LS-DYNA coupled field code.

Keyword Input

The LS-DYNA keyword input for the finite length wires model is shown in this section with the exception of the *node, *element and *set listings. Figure 15 summarizes the node, element and segment sets referenced in the keyword input.

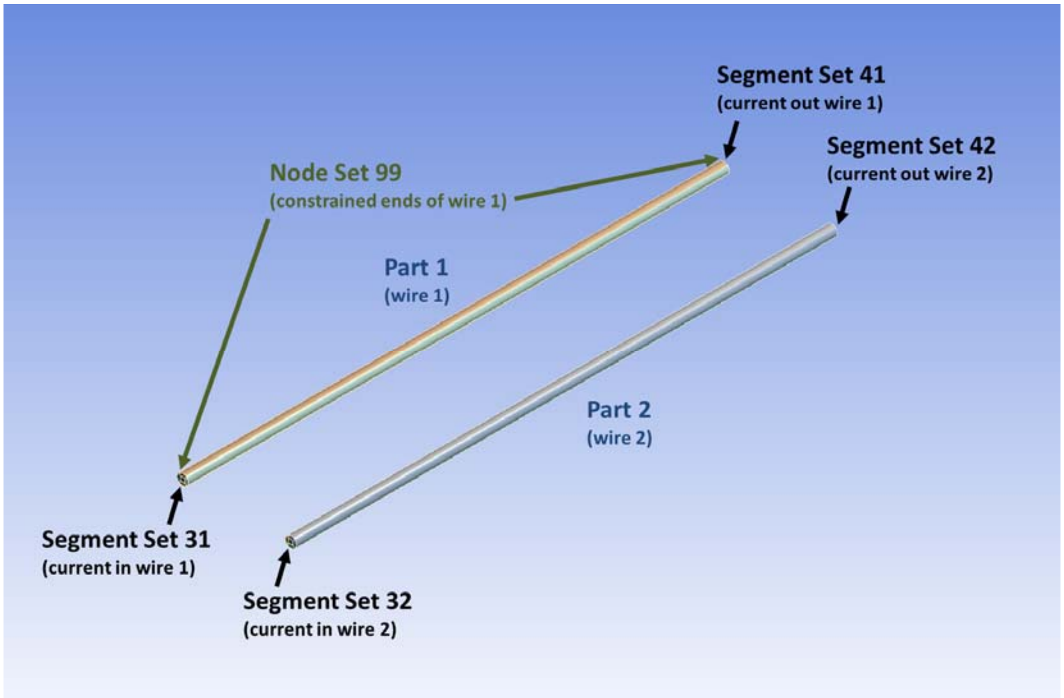
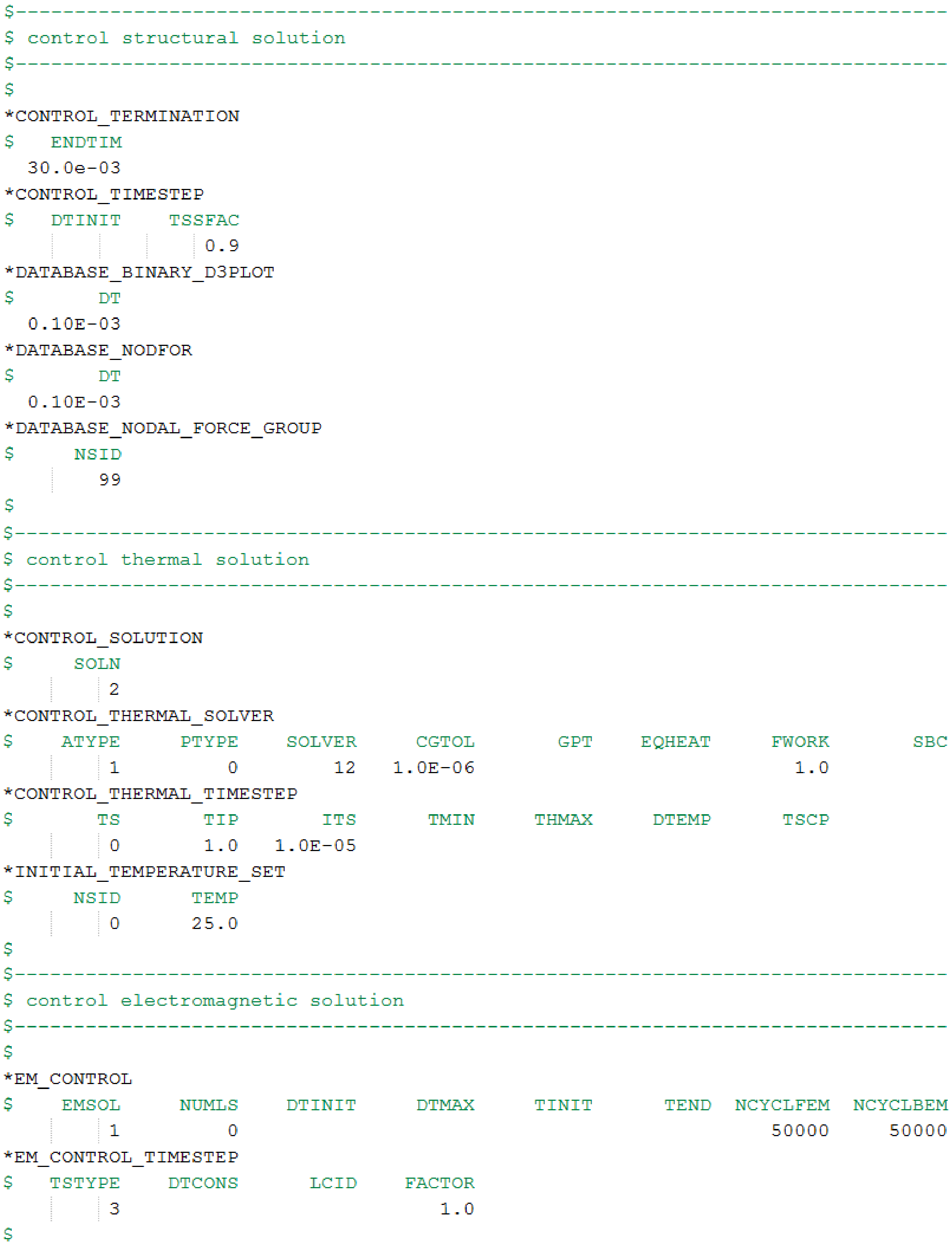
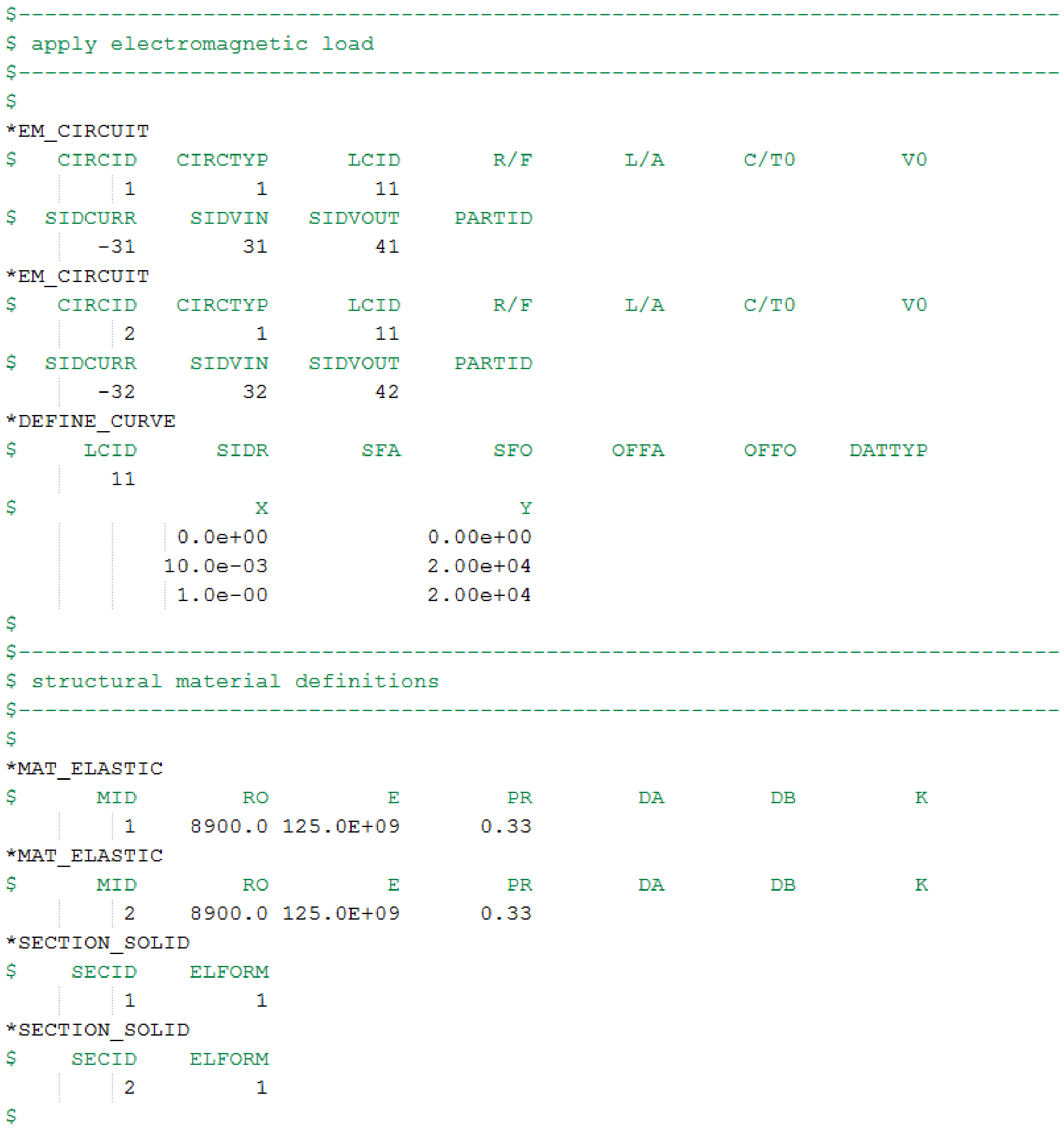


Figure 15. LS-DYNA Set and Part Numbers Used in Keyword Input

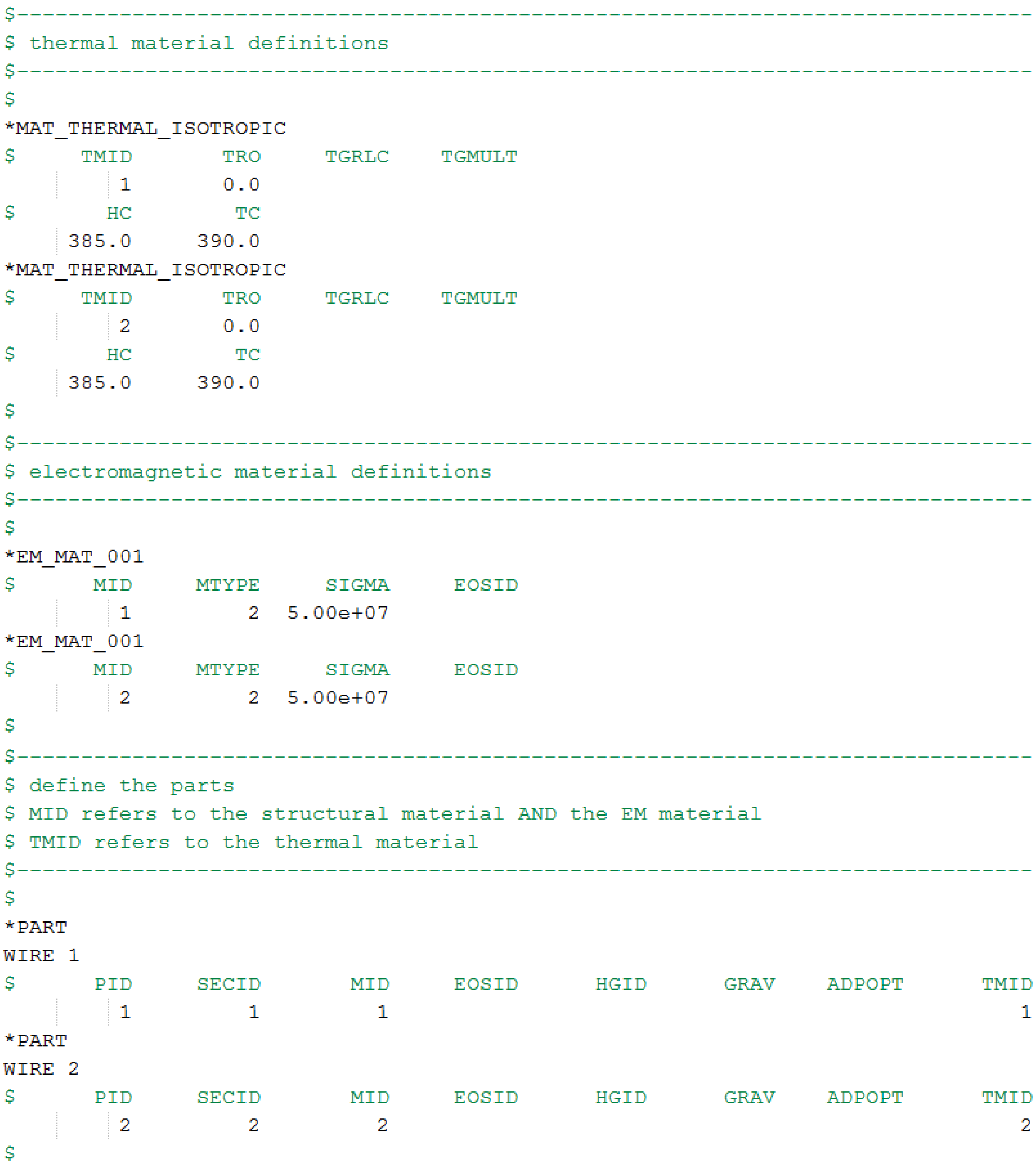
```
*KEYWORD
*TITLE
EM-Thermal-Structural Simulation of Parallel Wires Carrying Current
$
$-------------------------------------------------------------------------------
$ control mpp
$-------------------------------------------------------------------------------
$
*CONTROL_MPP_DECOMPOSITION_METHOD
RCB
*CONTROL_MPP_DECOMPOSITION_NUMPROC
         4
$
```



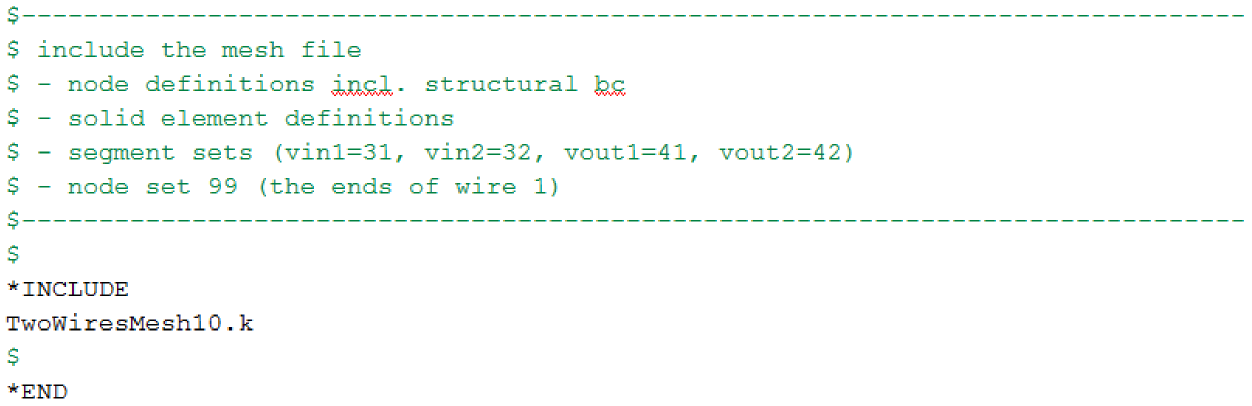
```
$-------------------------------------------------------------------------------
$ control structural solution
$-------------------------------------------------------------------------------
$
*CONTROL_TERMINATION
$   ENDTIM
  30.0e-03
*CONTROL_TIMESTEP
$   DTINIT    TSSFAC
                 0.9
*DATABASE_BINARY_D3PLOT
$       DT
  0.10E-03
*DATABASE_NODFOR
$       DT
  0.10E-03
*DATABASE_NODAL_FORCE_GROUP
$     NSID
        99
$
$-------------------------------------------------------------------------------
$ control thermal solution
$-------------------------------------------------------------------------------
$
*CONTROL_SOLUTION
$     SOLN
         2
*CONTROL_THERMAL_SOLVER
$    ATYPE     PTYPE    SOLVER     CGTOL       GPT    EQHEAT     FWORK       SBC
         1         0        12   1.0E-06                           1.0
*CONTROL_THERMAL_TIMESTEP
$       TS       TIP       ITS      TMIN     THMAX     DTEMP      TSCP
         0       1.0   1.0E-05
*INITIAL_TEMPERATURE_SET
$     NSID      TEMP
         0      25.0
$
$-------------------------------------------------------------------------------
$ control electromagnetic solution
$-------------------------------------------------------------------------------
$
*EM_CONTROL
$    EMSOL     NUMLS    DTINIT     DTMAX     TINIT      TEND  NCYCLFEM  NCYCLBEM
         1         0                                             50000     50000
*EM_CONTROL_TIMESTEP
$   TSTYPE    DTCONS      LCID    FACTOR
         3                             1.0
$
```



```
$-------------------------------------------------------------------------------
$ apply electromagnetic load
$-------------------------------------------------------------------------------
$
*EM_CIRCUIT
$   CIRCID   CIRCTYP      LCID       R/F       L/A      C/T0        V0
         1         1        11
$  SIDCURR    SIDVIN   SIDVOUT    PARTID
       -31        31        41
*EM_CIRCUIT
$   CIRCID   CIRCTYP      LCID       R/F       L/A      C/T0        V0
         2         1        11
$  SIDCURR    SIDVIN   SIDVOUT    PARTID
       -32        32        42
*DEFINE_CURVE
$     LCID      SIDR       SFA       SFO      OFFA      OFFO    DATTYP
        11
$                   X                   Y
             0.0e+00            0.00e+00
            10.0e-03            2.00e+04
             1.0e-00            2.00e+04
$
$-------------------------------------------------------------------------------
$ structural material definitions
$-------------------------------------------------------------------------------
$
*MAT_ELASTIC
$      MID        RO         E        PR        DA        DB         K
         1    8900.0 125.0E+09      0.33
*MAT_ELASTIC
$      MID        RO         E        PR        DA        DB         K
         2    8900.0 125.0E+09      0.33
*SECTION_SOLID
$    SECID    ELFORM
         1         1
*SECTION_SOLID
$    SECID    ELFORM
         2         1
$
```



```
$-------------------------------------------------------------------------------
$ thermal material definitions
$-------------------------------------------------------------------------------
$
*MAT_THERMAL_ISOTROPIC
$     TMID       TRO     TGRLC    TGMULT
         1       0.0
$       HC        TC
     385.0     390.0
*MAT_THERMAL_ISOTROPIC
$     TMID       TRO     TGRLC    TGMULT
         2       0.0
$       HC        TC
     385.0     390.0
$
$-------------------------------------------------------------------------------
$ electromagnetic material definitions
$-------------------------------------------------------------------------------
$
*EM_MAT_001
$      MID     MTYPE     SIGMA     EOSID
         1         2  5.00e+07
*EM_MAT_001
$      MID     MTYPE     SIGMA     EOSID
         2         2  5.00e+07
$
$-------------------------------------------------------------------------------
$ define the parts
$ MID refers to the structural material AND the EM material
$ TMID refers to the thermal material
$-------------------------------------------------------------------------------
$
*PART
WIRE 1
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1                                                 1
*PART
WIRE 2
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2                                                 2
$
```



```
$-------------------------------------------------------------------------------
$ include the mesh file
$ - node definitions incl. structural bc
$ - solid element definitions
$ - segment sets (vin1=31, vin2=32, vout1=41, vout2=42)
$ - node set 99 (the ends of wire 1)
$-------------------------------------------------------------------------------
$
*INCLUDE
TwoWiresMesh10.k
$
*END
```