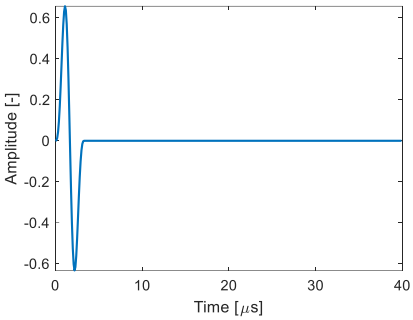
Finite Element Method using MSC.Marc - Dynamics (wave propagation)

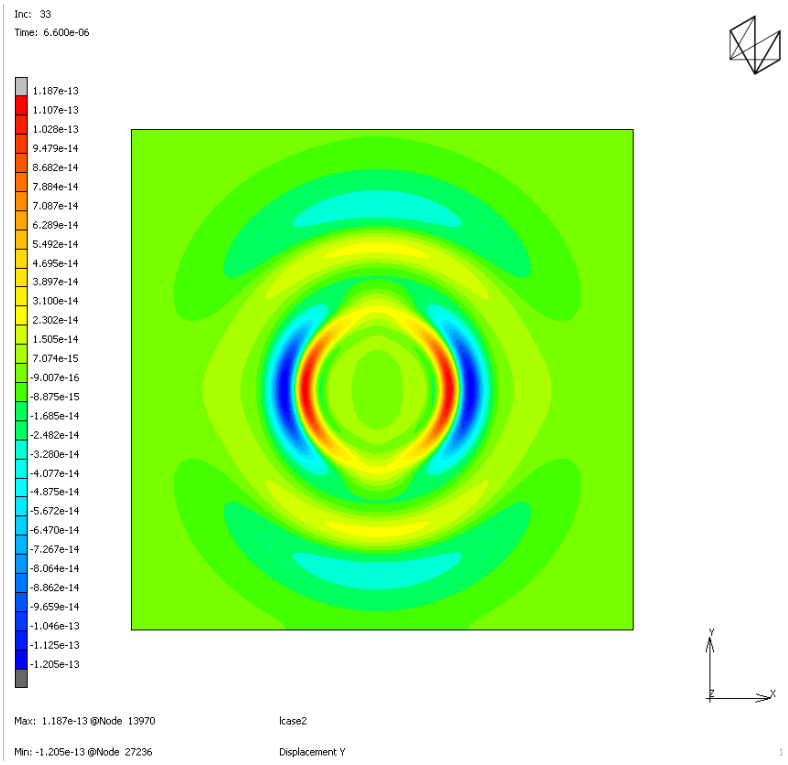
The purpose of this lab is to practice performing transient simulations to analyze dynamic phenomena, using elastic wave propagation as an example.

Test case no. 1

1. Create a 2D square plate (***Planar analysis***) with dimensions of **0.1 x 0.1 meters**. Discretize your model using quadrilateral (***Quad 4***) elements with a size of **0.5 x 0.5 millimeters**. Specify the material properties for aluminum.
2. Create an excitation signal in the time domain with a central frequency of **1 MHz**, a sampling frequency of 20 MHz, and one cycle. See the example on the right. You can use MATLAB to create the signal. Once the excitation signal is created, copy the time and amplitude vectors into a .txt file and name it *excSignal_1MHz*, for example.

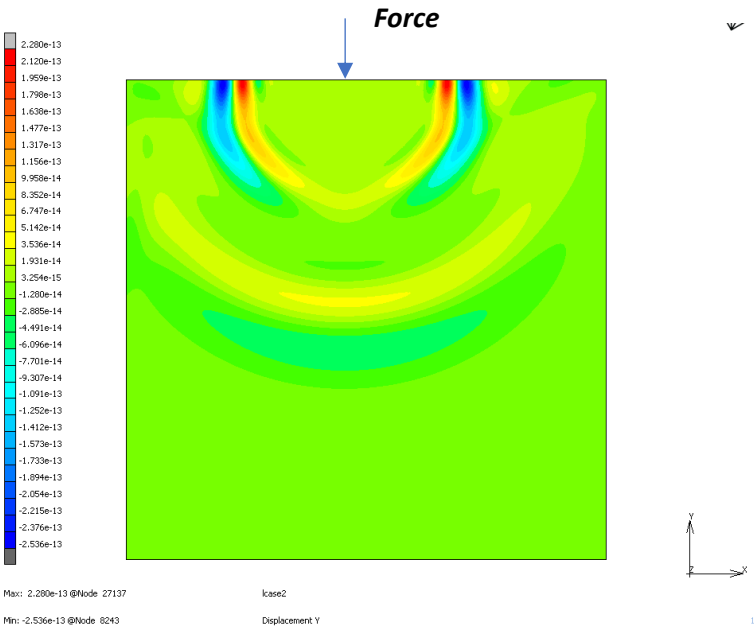


3. Import your excitation signal into Marc by going to ***Tables & Coord. Syst.*** tab and select ***Read*** to load your excitation signal. In the ***Table Properties*** window, select ***Independent variable V1*** type as ***Time.*** In addition, make sure to uncheck (deactivate) the ***Extrapolate*** option.
4. Assign a boundary condition of ***Point Load*** in the X or Y direction in the middle of the plate using the previously created signal.
5. Create a ***Dynamic Transient*** load case and run your simulation for **20 µs** with a constant time step of **0.2 µs**.
6. Once the simulation is finished, open the results and plot a history plot for the selected node. A tip on how to create a history plot can be found at the end of the instructions.



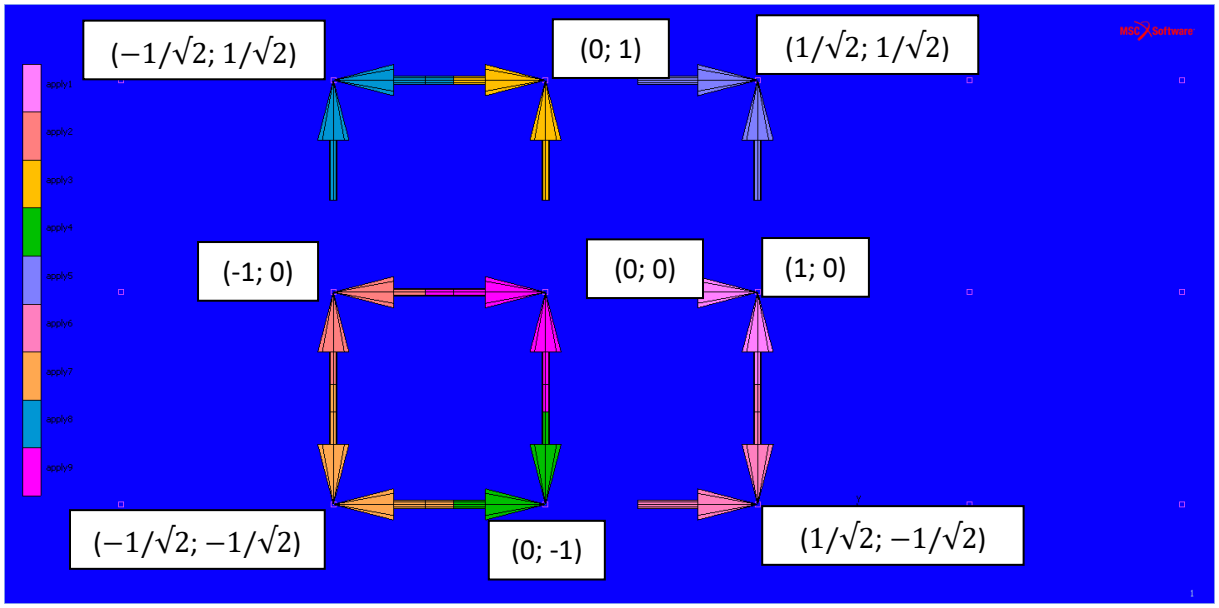
Test case no. 2

Move your boundary condition to the top surface of your model, then run the simulation. Similar to the example shown below, guided and bulk waves should now be observed.



Test case no. 3

Repeat the test case no. 1 but apply the boundary conditions in a manner similar to that presented below.



Compare the results from task no. 1 with those from task 3.

Test case no. 4

Add another material or discontinuity to the previous model in the selected area and run the calculations. Then, compare the results and explain the differences.

How to make a history plot:

Results tab – ***General*** – ***History Plot*** – ***Set Location*** (select nodes for which you would like to make a plot) – ***All Incs*** – ***Add Curves***. In the *Add History Plot Curves* window select options as shown below:

