Application Note

Piezoelectric Fluid-Structure Interaction

Application Note: Piezoelectric Fluid-Structure Interaction
Version 8/PC

Part Number 30-090-101
September 2008

Printed in the United States of America

Patent Number 6,116,766: Fabrication Based Computer Aided Design System Using Virtual Fabrication Techniques
Patent Number 6,157,900: Knowledge Based System and Method for Determining Material Properties from Fabrication and Operating Parameters

Table Of Contents

1 Introduction

This example models a piezoelectric pump, a typical fluid-structure interaction case, using the ThermoElectroMechanical analysis module of IntelliSuite. The device consists of an electrode pair surrounding a piezoelectric film and a liquid chamber beneath with two entry ports.

When a sinusoidal voltage load is applied between the electrodes, the piezoelectric material deforms, causing flow through the pump. By controlling the voltage, one can control the flow rate and pressure of the pump.

Click *Start…Programs…IntelliSuite…ThermoElectroMechanical*

The ThermoElectroMechanical analysis module will appear. The model that will be used for this analysis has already been created.

Click *File…Open*

Select SquarePiezoPump.save from the \IntelliSuite\Training\piezo_pump directory.

Click *Open*

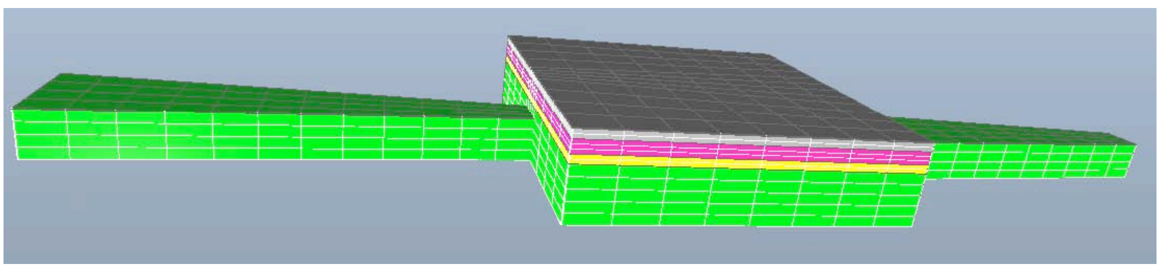

Figure 1: Piezoelectric pressure sensor model

2 Simulation Setting

Click *Simulation…Simulation Setting*

Set the *Calculation Type* to *Dynamic* and the *Analysis Type* to *Stress/Disp.*. In the *Piezo Material* section of the *Option* box, check the box next to *Piezoelectric (undeformed shape only)*. In the *Dynamic* section, check *Transient (Fixed Time Increment)*, and set the *Time Period* 0.1, and *Increment Number* 200, as shown in Figure 2.

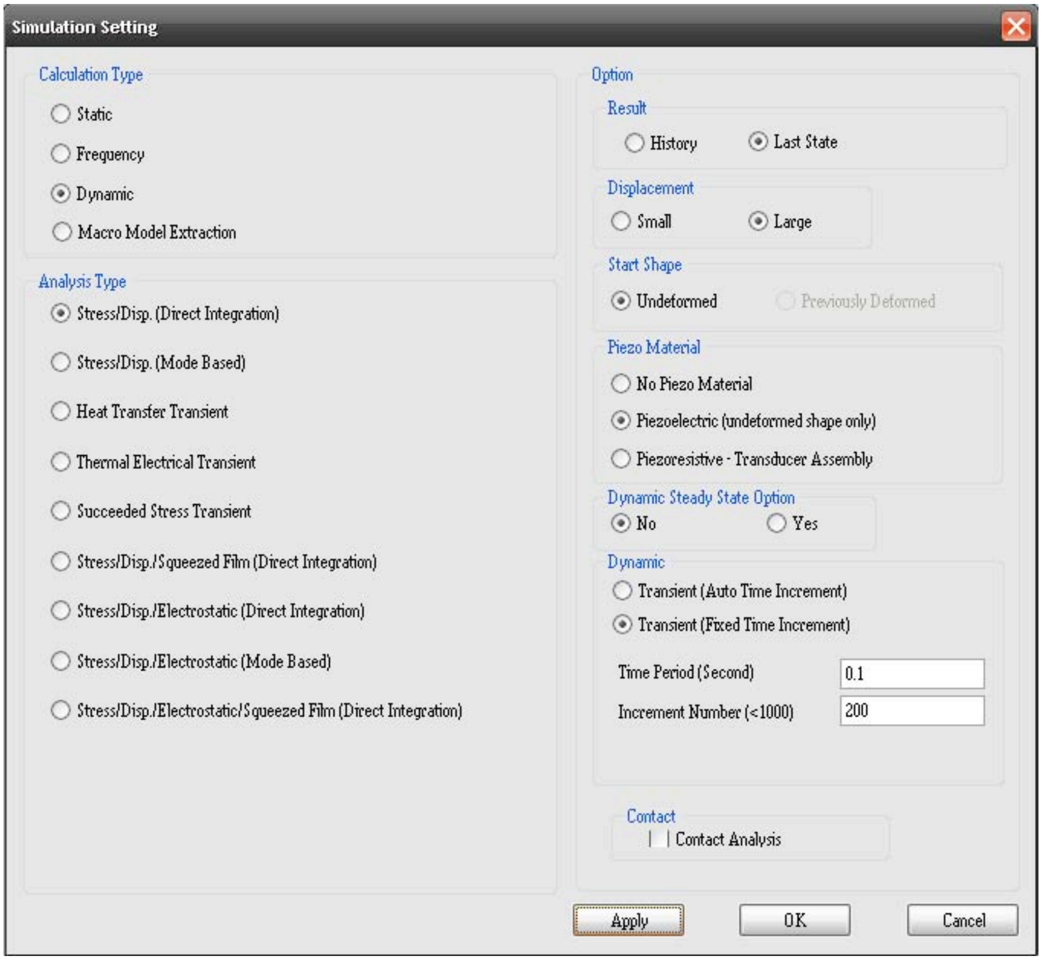

Figure 2: Simulation Setting dialog box

Click *OK*

3 Material Properties

The piezoelectric material must be defined and the value of damping must be adjusted.

Click *Material…Define Piezoelectric Material*

Select the pink entity, which will turn red as shown in Figure 3.

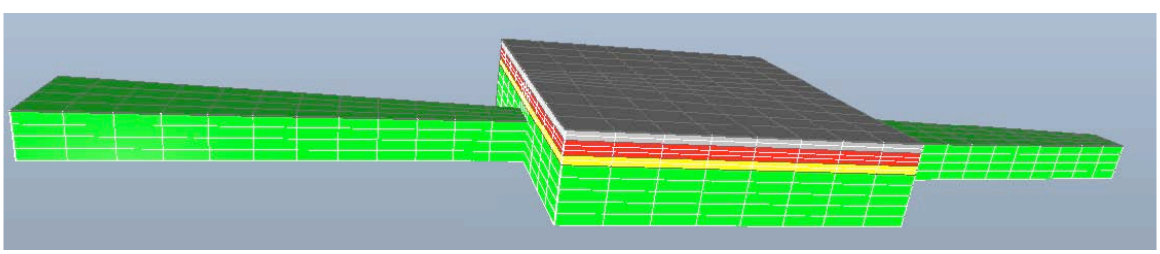

Figure 3: Selected piezoelectric material (highlighted in red)

The *Define Piezoelectric Strain Coefficient Matrix* dialog box will appear as shown in Figure 4.

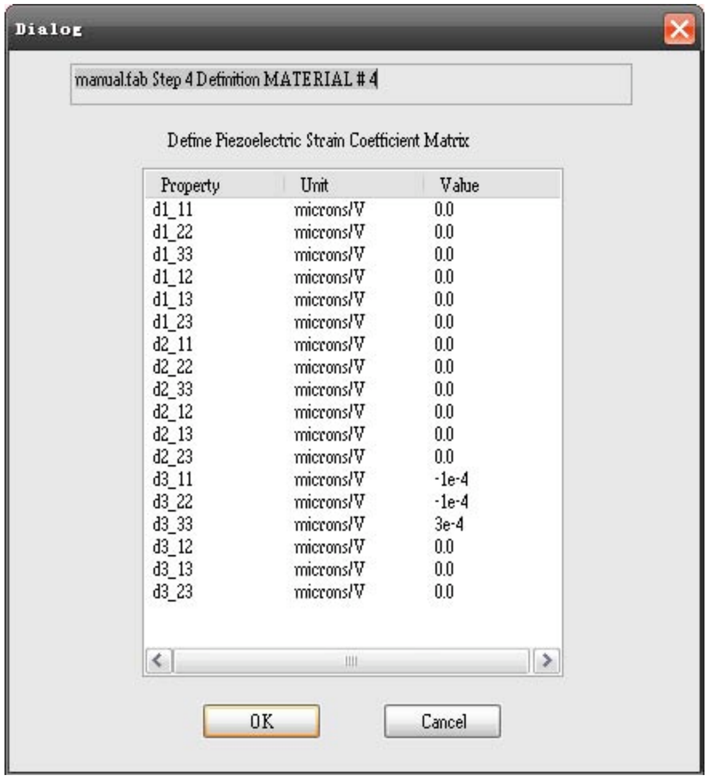

Figure 4: Piezoelectric properties dialog box

Leave the values as they are.

Click *OK*

Click *Material…Damping Definition*

Select the yellow entity, shown in red in Figure 5.

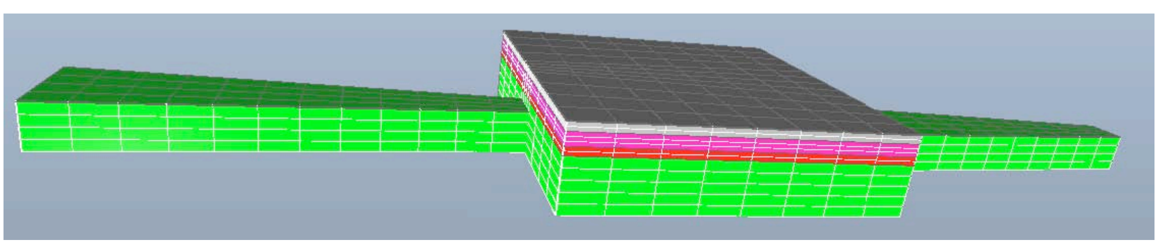

Figure 5: Defined damping material

The *Check/Modify Material Property* dialog box will appear as shown in Figure 6.

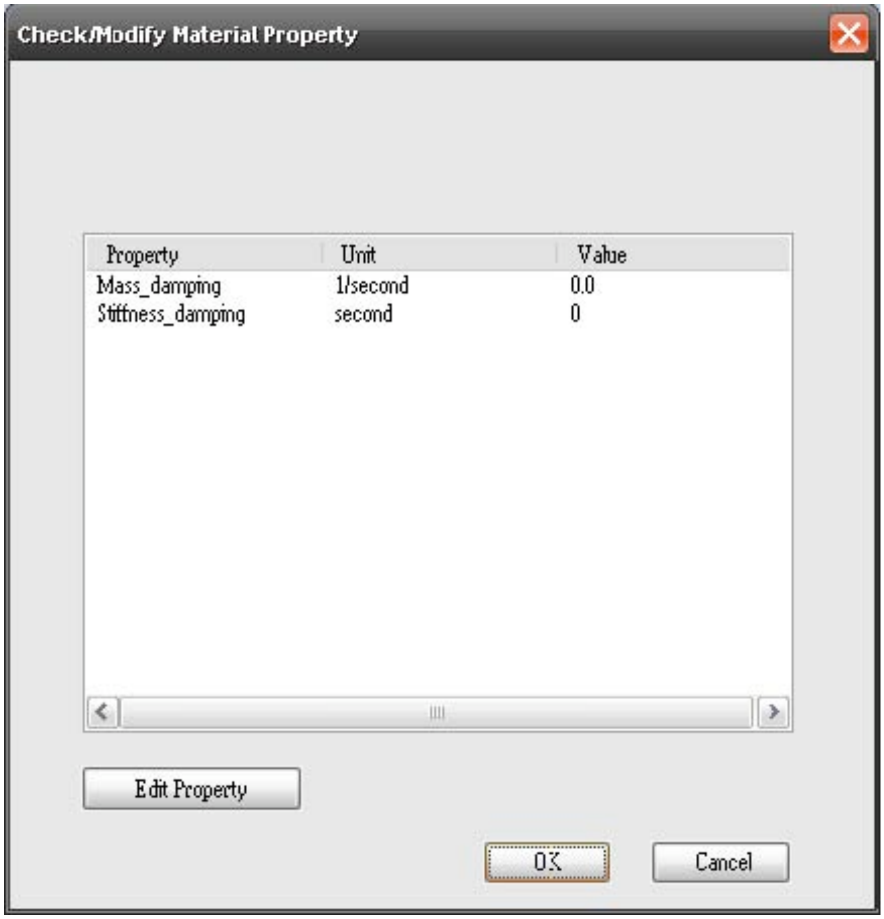

Figure 6: Damping properties dialog box

Leave the Mass_damping and Stiffness_damping at 0. For fluid-structure interaction simulations, the fluid will be the major source damping and we do not need to define any other damping.

Click *OK*

Click *Define/Modify Fluid Entity*

Select the light-green entity, and the *Fluid properties* dialog box will appear.

Click *Add as Fluid entity*

Leave the Viscosity as 0.001 and the Density as 1000 (shown in Figure 7).

Click *OK*

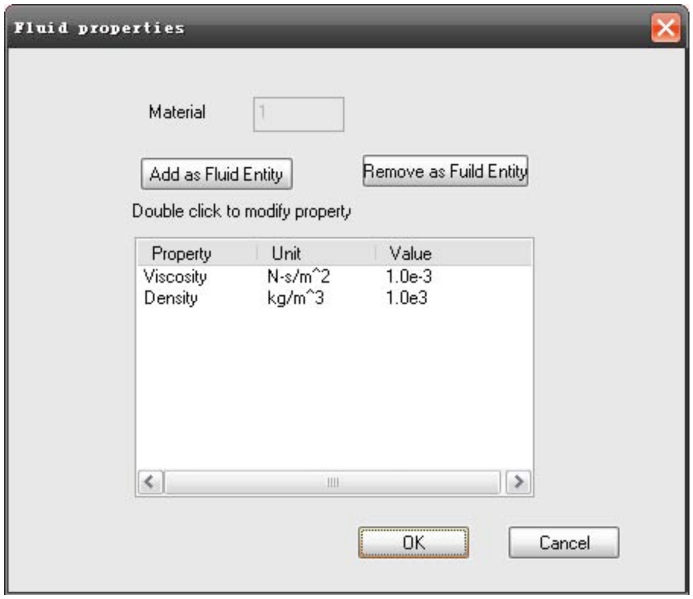

Figure 7: Fluid properties dialog box

4 Loading Conditions

Click *Loads…Amplitude vs Time…Periodic*
Click *Loads…Voltage…Entity*

Select the gray entity and set the voltage as in Figure 8.

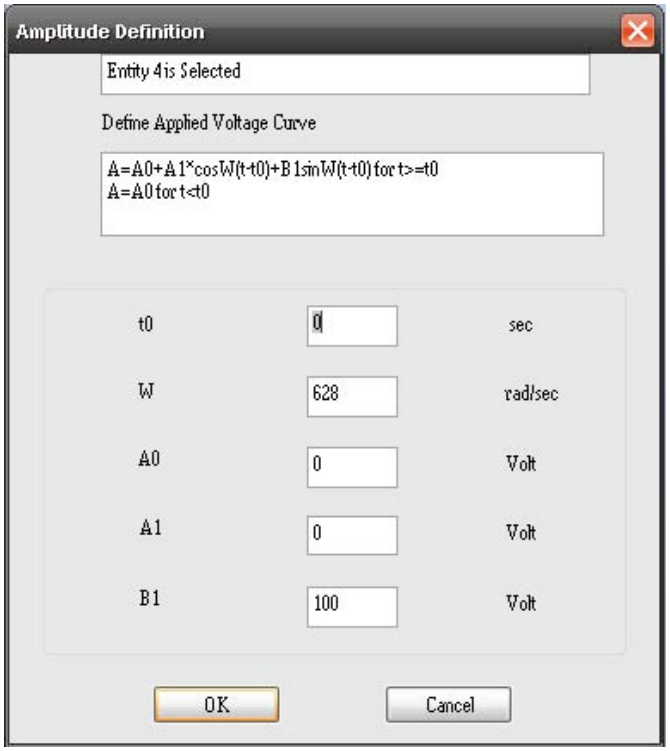

Figure 8: Voltage setting dialog box

Select the yellow entity, and leave its voltage at zero.

5 Boundary Conditions

The four lateral faces of the membrane (shown in red in Figure 9) must be anchored.

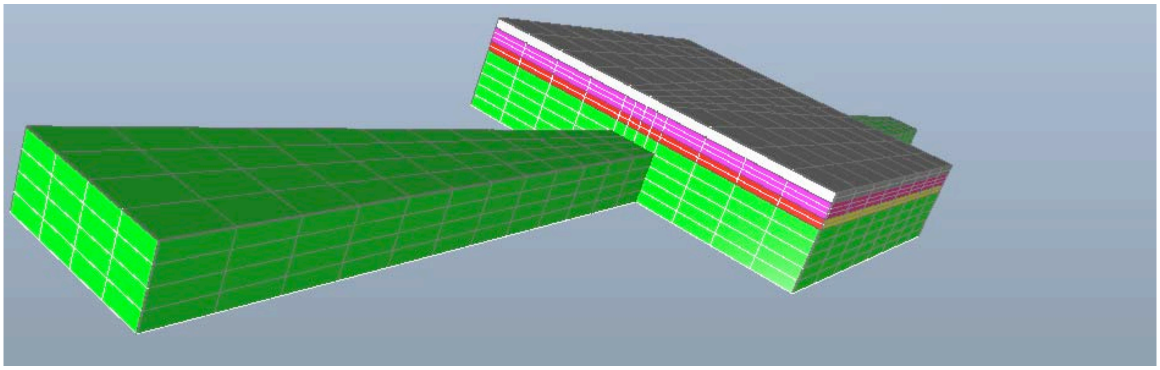

Figure 9: One fixed face of membrane

Click *Boundary…Selection Mode…Check Only*

Click *Boundary…Fixed*

The anchored faces have the "Fixed" boundary already applied. They will highlight in red, as shown in Figure 9.

The two fluid ports must be defined.

Click *Boundary…YZ Fixed*

The fluid ports will become highlighted in red as shown in Figure 10 and Figure 11. *YZ fixed* means that the fluid velocity in the Y and Z directions is held zero. The simulation will solve for the velocity in the X direction on the selected faces.

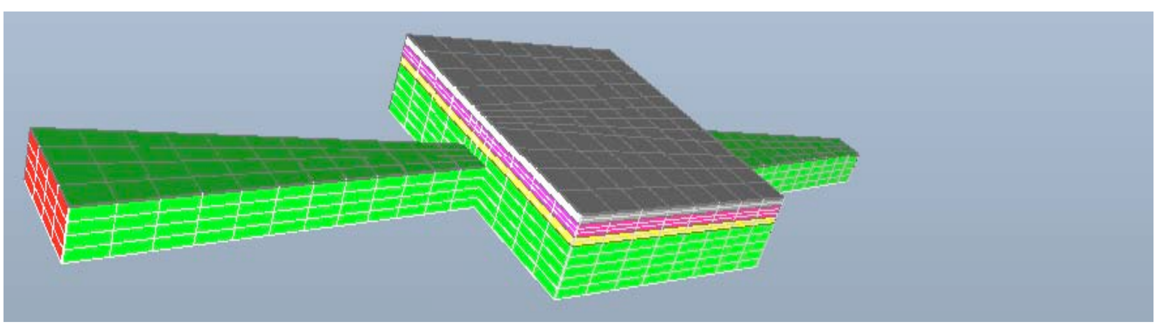

Figure 10: First fluid port

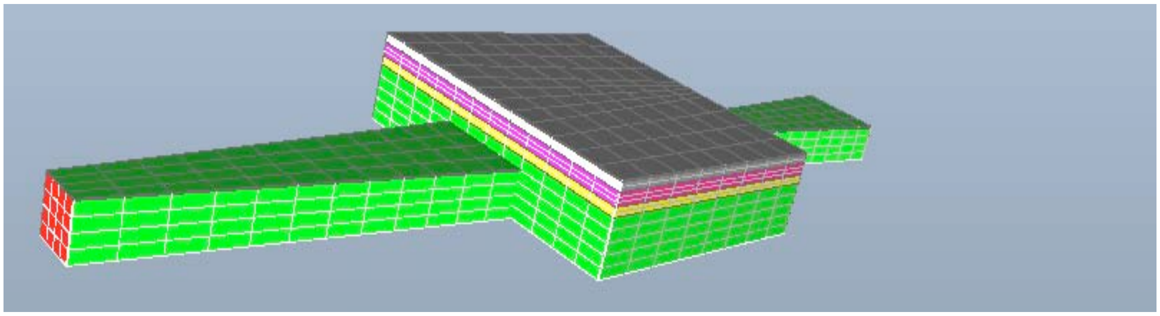

Figure 11: Second fluid port

6 Dynamic Analysis

Start the analysis.

Click *Analysis…Start Dynamic Analysis*

A Fluid Dynamic Computation dialog box will appear (Figure 12). Leave the values at the default.

Click *OK*

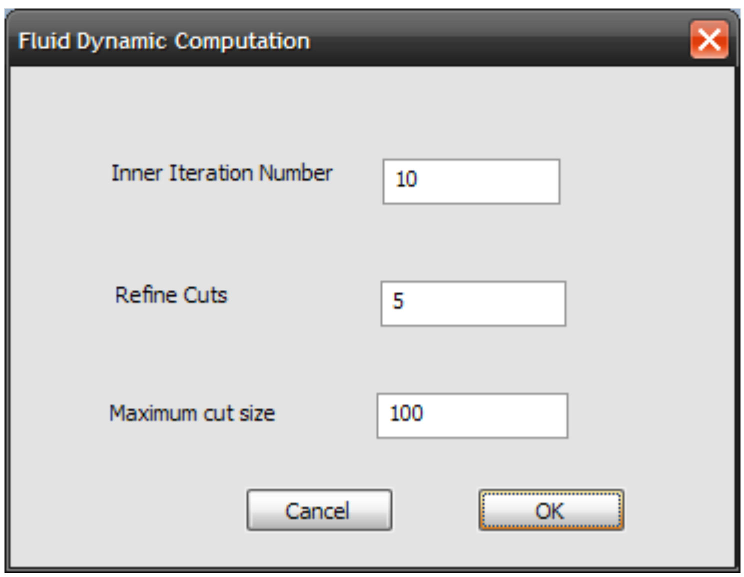

Figure 12: Fluid Dynamic Computation settings

7 Results

A command prompt will appear. It will look something like Figure 13.

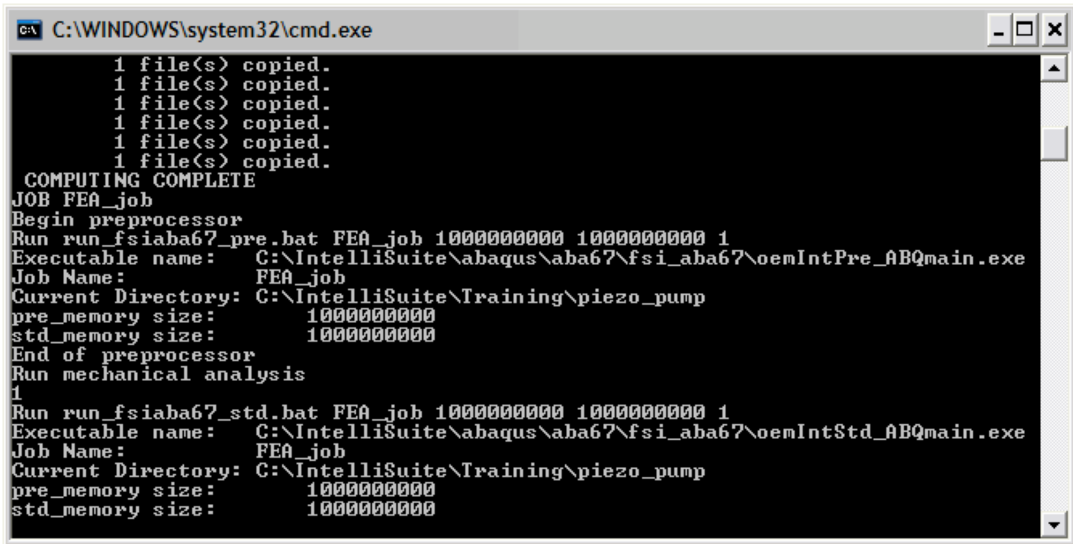

Figure 13: Analysis command prompt

When the command prompt disappears, the simulation is complete. We are interested in a few of the results.

Click *Result…2D Plot, Mechanical Analysis…Maximum…Dis_Z(Mag)*

The displacement is sinusoidal function of time (Figure 14).

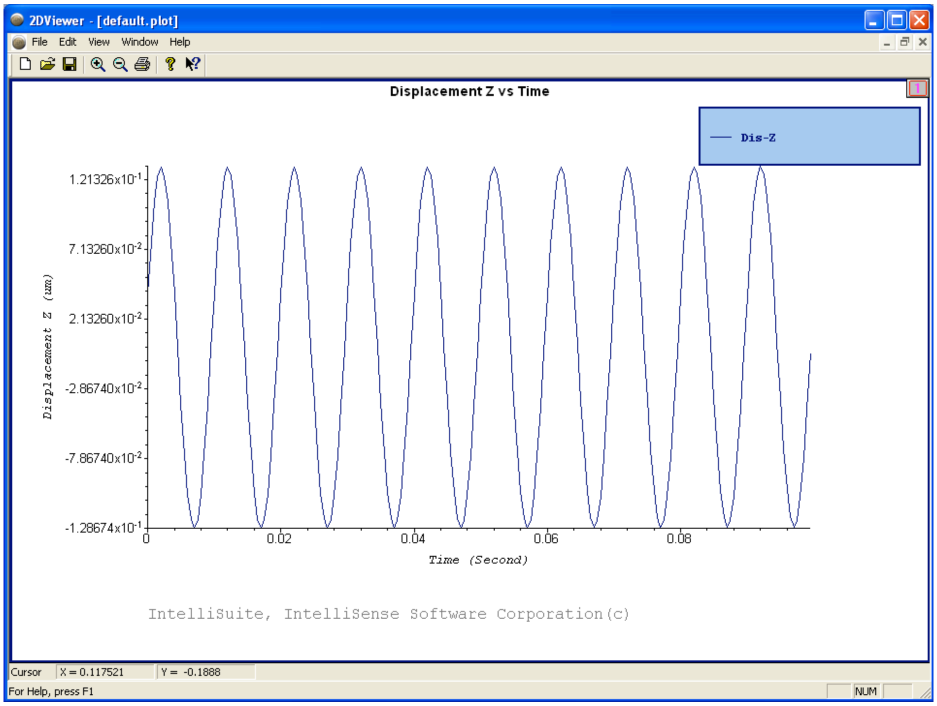

Figure 14: Dynamic mechanical response

In the 2D Viewer window, open the *flowrate.plot* file, which will be in the same directory as the current *.save file. The flow rate can be obtained by calculating the slope of the line shown in the figure below.

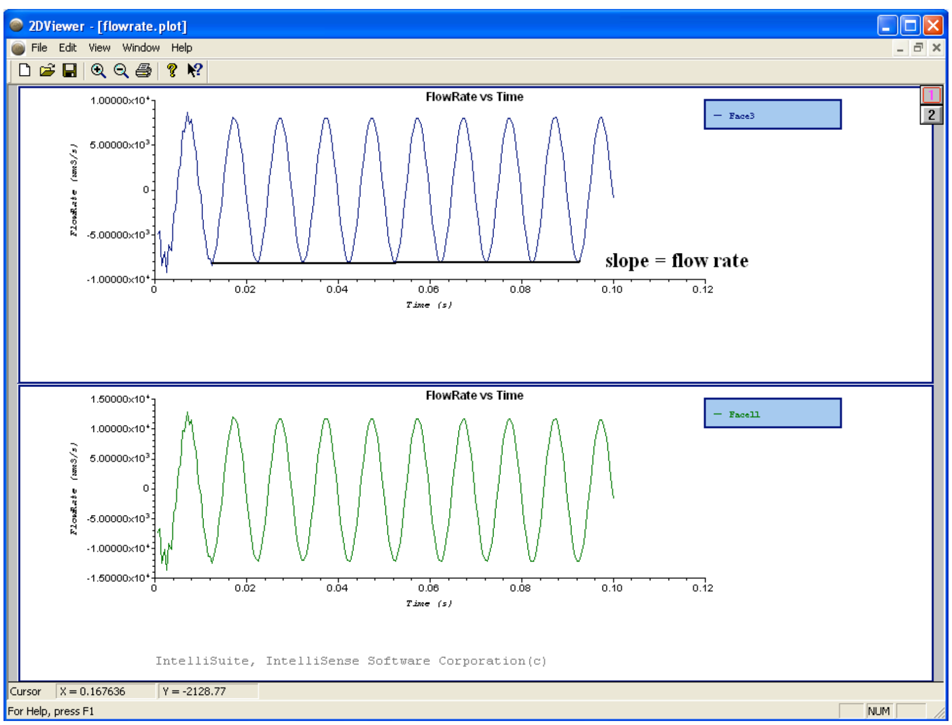

Figure 15: Flow rate vs. time

The rest of the results will be more easily evaluated in VisualEase, the post-processing module in IntelliSuite.

Click *Analysis…Open in VisualEase*

Select *dump_001.plt*. VisualEase will open with the model. Deselect *Surface* and select *Vector* in the menu on the left. The Vector Settings dialog will open. Set up the vectors as shown below.

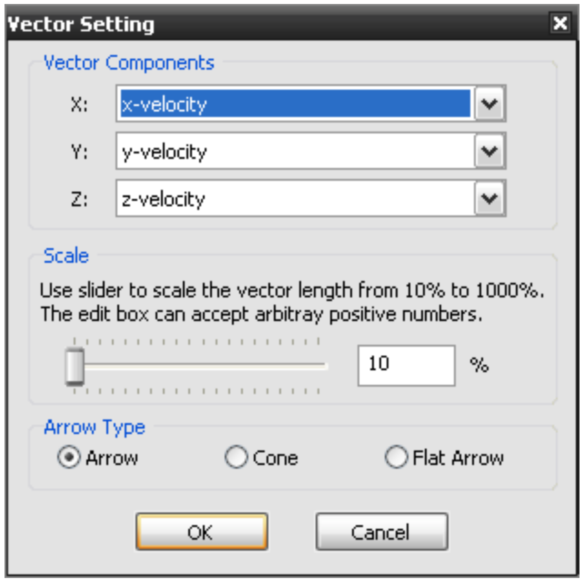

Figure 16: Vector Settings

The model will appear as shown below. Use the play button to view an animation of the simulation.

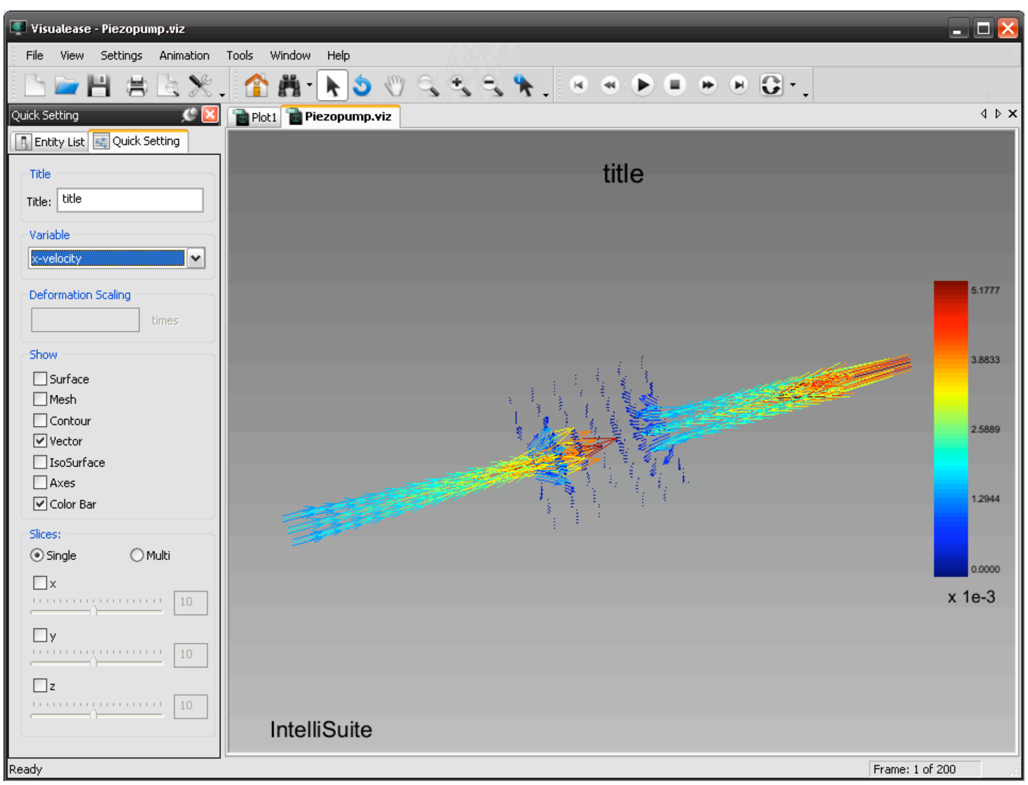

Figure 17: Piezopump model in VisualEase