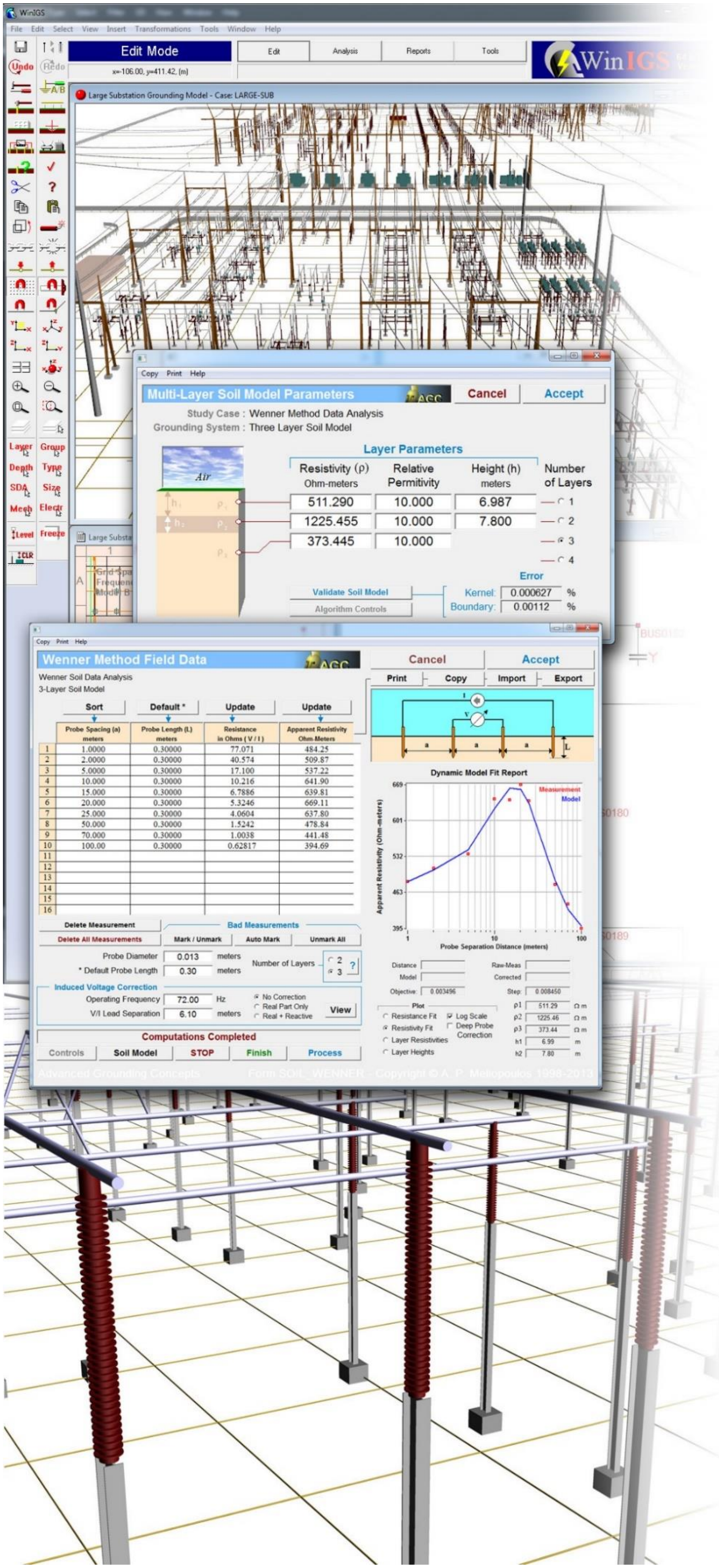


WinIGS

Windows Based **I**ntegrated **G**rounding **S**ystem Design Program

Structural **D**ynamic **A**nalysis Training Guide

Program Version 7.6
Last Revision: September 2020

NOTICES

Copyright Notice

Disclaimer

The developer is neither responsible nor liable for any conclusions and results obtained through the use of the program WinIGS.

Contact Information

For more information concerning this program please contact:

Advanced Grounding Concepts
P. O. Box 49116
Atlanta, Georgia 30359,

Telephone: 1-404-325-5411, Fax: 1-404-325-5411
Email: sakis@comcast.net

Table of Contents

Structural Dynamic Analysis Training Guide Overview

The program WinIGS is an analysis/design tool for multiphase power systems. The program follows an integrated modeling approach which includes detailed models of most major power system components such as transmission lines transformers generators, grounding systems, etc. All application tools (grounding analysis, fault current analysis, structural dynamic analysis, lightning shielding analysis, etc.) are based on an integrated physical model. The advantage of this approach is that once the integrated model has been developed, all application tools can be applied without additional user effort.

This training guide focuses on the structural dynamic analysis of buswork and bus support structures with the following objectives:

- Familiarize the user with the structural dynamic analysis program features, user interface, and usage procedures.
- Illustrate the capabilities of the structural dynamic analysis tools.
- Compare the SDA analysis results with results from computational methods outlined in the IEEE Standard 605.

The model data files associated with each application example are included in WinIGS program setup. The user is encouraged to experiment with these examples by modifying the system data, as well as the analysis parameters, executing various analysis functions and studying the analysis reports.

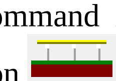
1. Using The WinIGS SDA Models and Tools

1.1 Introduction

The WinIGS Program includes a number of models and tools that can be used to perform Structural Dynamic Analysis of substation buswork. Both rigid and flexible bus structures can be analyzed. This section describes the use of these models and tools.

The SDA models and tools are available within the WinIGS geometric grounding model. While in editing any geometric grounding model the following SDA specific models and tools can be added to the model:

1. Conductor (SDA)
2. Structural Beam (SDA)
3. Support Element (SDA)
4. Mechanical Connector (SDA)
5. Mechanical Source (SDA)
6. Structural Dynamics Meter (SDA)

All of the above elements are created using the command ***Insert SDA & Buswork Elements*** of the ***Insert*** menu, or using the toolbar button .

Structures to be analyzed are modeled using the first four elements listed above. Specifically, all bus support structures, such as beams as well as insulators are represented using the *SDA Structural Beam Model*. Electrical conductors including both rigid and flexible are represented using the *SDA Conductor* element. Non-rigid joints among structural elements or conductors which allow for sliding or rotation are represented using the *SDA Mechanical Connector* model. Finally, any SDA model structure must include *SDA Support Elements*, which represent the mechanical constrains provided by the foundations supporting the modelled structure.

Structural Dynamic Analysis models can be subjected to a number of static and dynamic mechanical “loads”, such as (1) Gravity, (2) Magnetic Forces, (3) Wind & Ice Loading, and (4) Earthquake Motion. Furthermore a *Mechanical Source Model* is provided which can apply a user defined concentrated force or moment at any point of the modelled structure.

The Structural Dynamic Meter element provides a flexible method of reporting the results of the structural dynamic analysis. Specifically, any number of meters can be added to user selected locations. Each meter generates time histories of displacements, rotations, forces moments and stresses at the selected locations. These time histories can be presented in a plot format, and also written into a standard CSV file (which can be viewed using a spreadsheet program such as Microsoft Excel). Furthermore, reports of the maximum and final values of these time histories can generated.

Detailed descriptions of the SDA elements are presented in the next Sections.

1.2 The SDA Conductor Element

The SDA conductor element can be used to represent both rigid and flexible conductors. The model captures both conductor electrical and mechanical properties. Entering the conductor geometry is accomplished by defining the conductor end points. Any number of points can be added resulting in a “polygonal” element. Points are added using the mouse in top-view mode (x-y). Note that during the end point entry process, the z-coordinates are initially set to a default value. Upon completion of the end point entry, the z-coordinates can be modified either by numerical entry (using the properties window) or graphically (using the mouse) after switching to a side view (either XZ, or YZ view).

Once the end points of a conductor model have been defined, the conductor type and size must be selected. The selections is made from the conductor properties window by accessing a set of conductor libraries. To open the conductor properties dialog, left double click on the conductor image (See Figure 1.1).

To select the conductor Type & Size click on the entry fields in the block titled Section Type and Size.

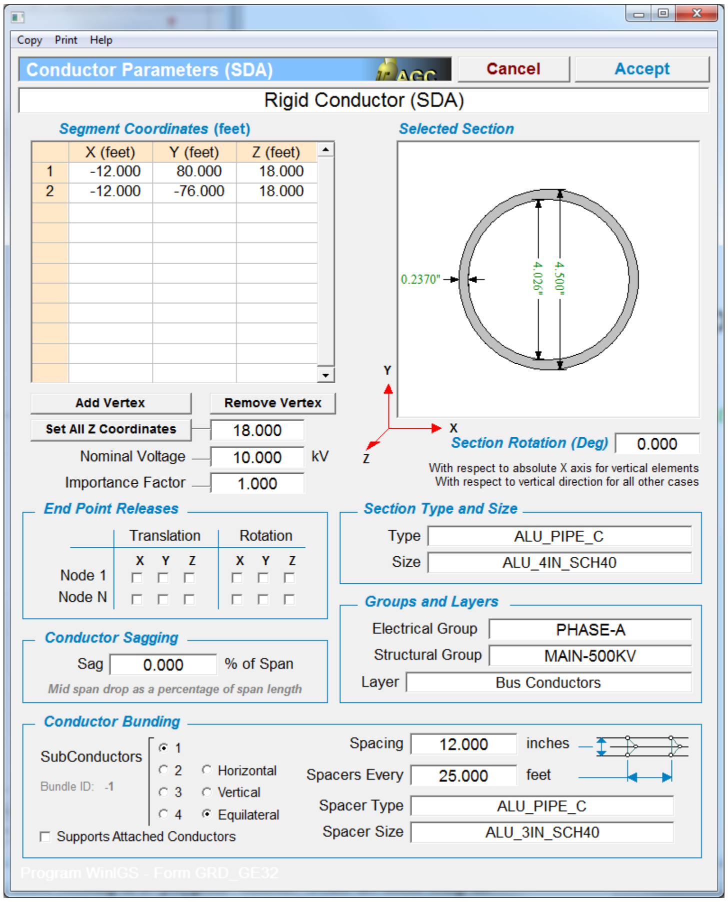


Figure 1.1: Conductor Parameters Dialog Window

1.3 The SDA Structural Beam Element

The SDA beam element can be used to represent bus supporting structures consisting of various types of steel or aluminum beams, as well as insulator structures. The model simulates the structure mechanical properties. Entering the beam geometry is accomplished by defining the beam end points and the cross-section orientation. Any number of points can be added resulting in a “polygonal” element. Points are added using the mouse in top-view mode (x-y). Note that during the end point entry process, the z- coordinates are initially set to a default value. Upon completion of the end point entry, the z-coordinates can be modified either by numerical entry (using the properties window) or graphically (using the mouse) after switching to a side view (either XZ, or YZ view).

Once the end points of a beam element have been defined, the beam type and size must be selected. The selections is made from the beam properties window by accessing a set of section libraries. To open the properties dialog, left double click on the element image (See Figure 1.2).

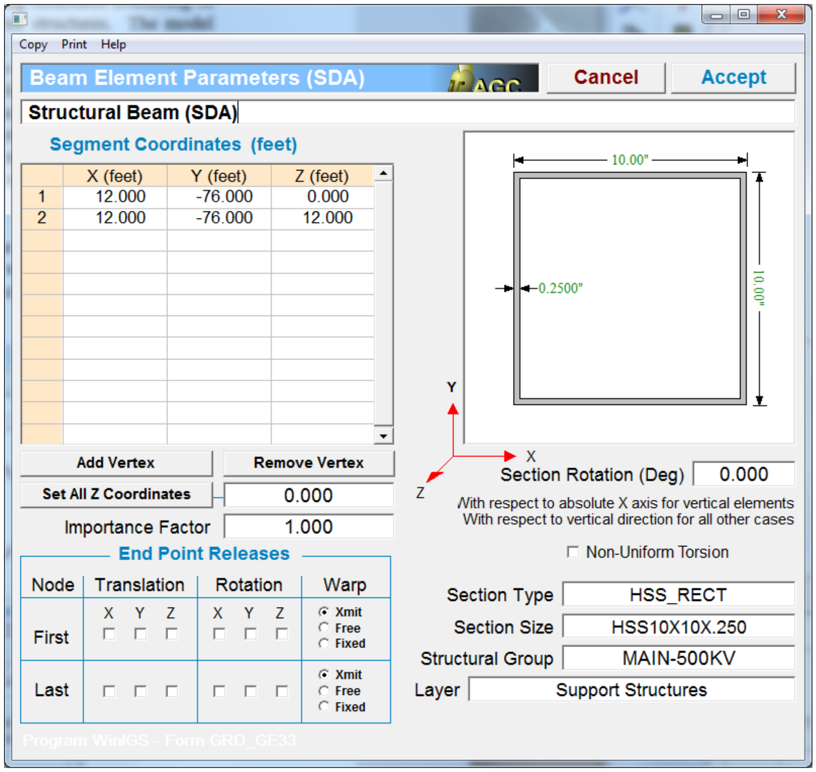


Figure 1.2: Beam Parameters Dialog Window

The section orientation is defined by specifying the Section Rotation Angle (in Degrees). The beam end point translation and rotation release controls allow the beam ends to rotate or translate freely along the major global axes. By default, all releases are turned off, making the beam ends solidly connected to any other coincident beams.

To select the beam type & Size left-click on the entry fields titled Section Type and Section Size, to open the selection window (illustrated in Figure 1.3). The section type and size selection window allows selection of 25 different section types (such as I beams, L and C beams, pipes, rectangular beams, etc.) Upon selection of the beam type from the left column of the window, the corresponding available sizes appear of the right column. Upon selection of both desired type and size the corresponding mechanical properties of the beam section are displayed within the Properties field group, and the cross-section image is displayed.

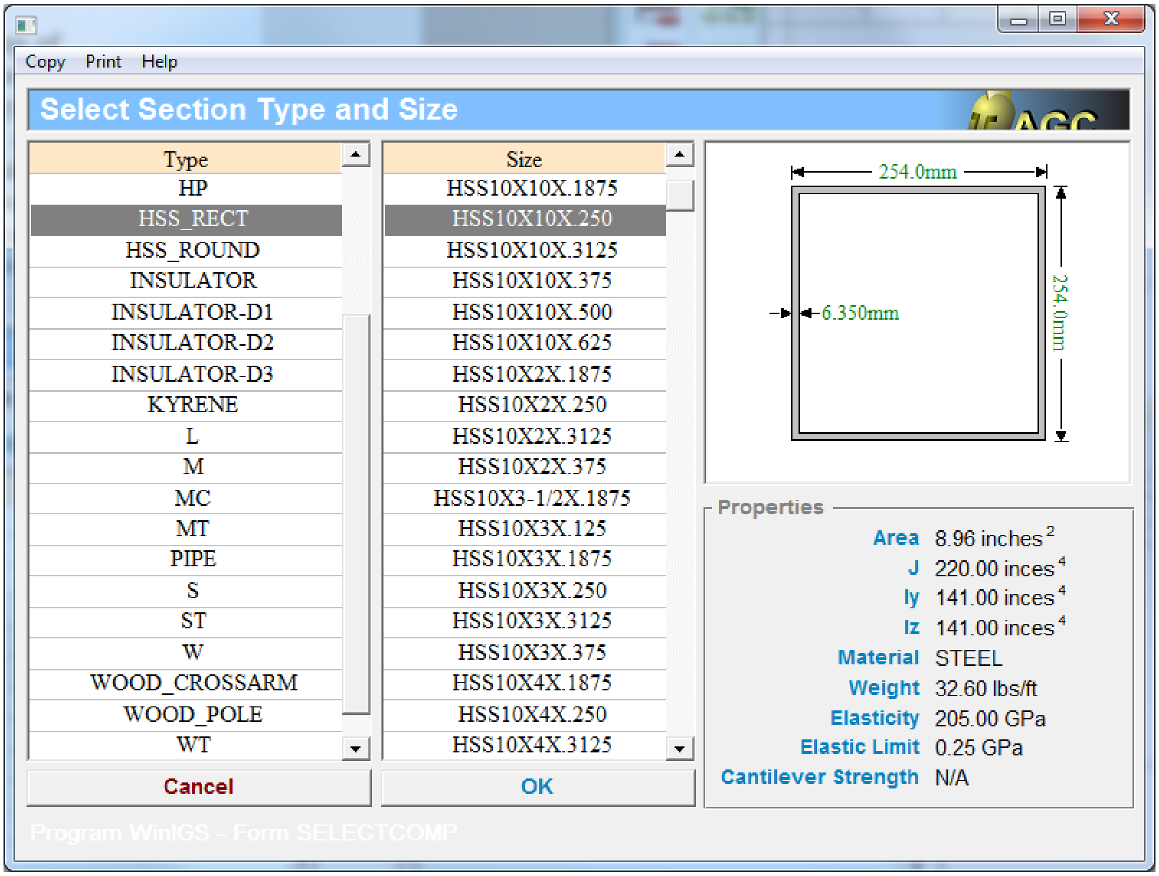


Figure 1.3: Beam Section Type and Size Selection Dialog Window

1.4 The SDA Support Element

The SDA support element enforces translation and rotation boundary conditions at a selected point. It is typically used to model the connection of structures consisting of beam elements to the foundation. A support element example is shown in Figure 1.4a.

Support elements are added using the mouse in top-view mode (x-y) using a single left-click at the desired location. Note that during the entry process, the z- coordinate is initially set to zero The z-coordinate can be later modified either by numerical entry (using the properties window) or graphically (using the mouse) after switching to a side view (either XZ, or YZ view).

The SDA properties window (opened by a left double click on the support element image) is illustrated in See Figure 1.4

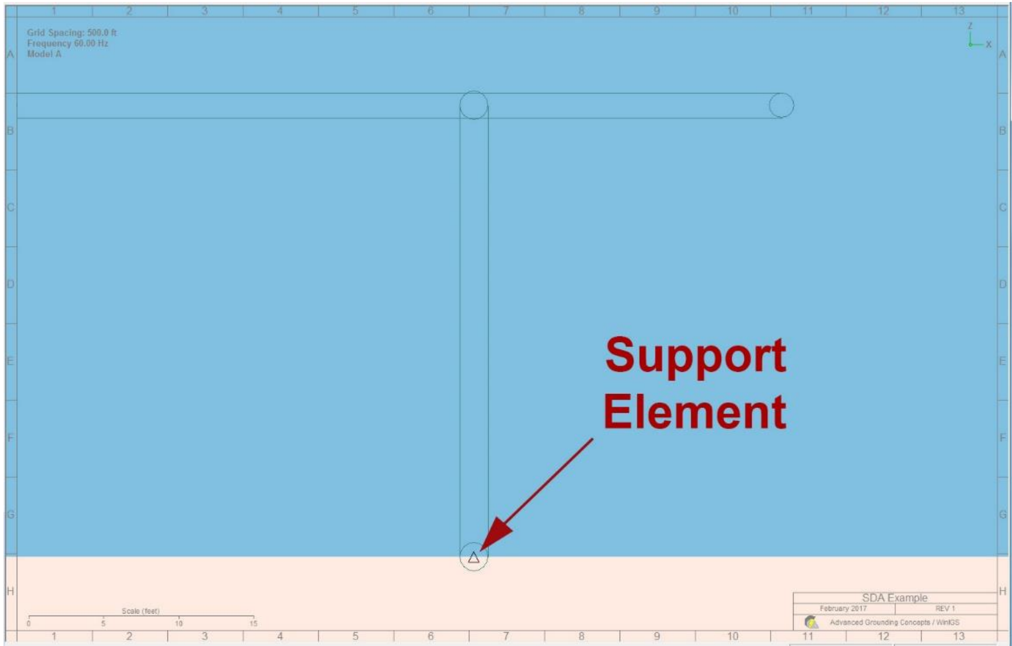


Figure 1.4a: Support Element Parameters Dialog Window

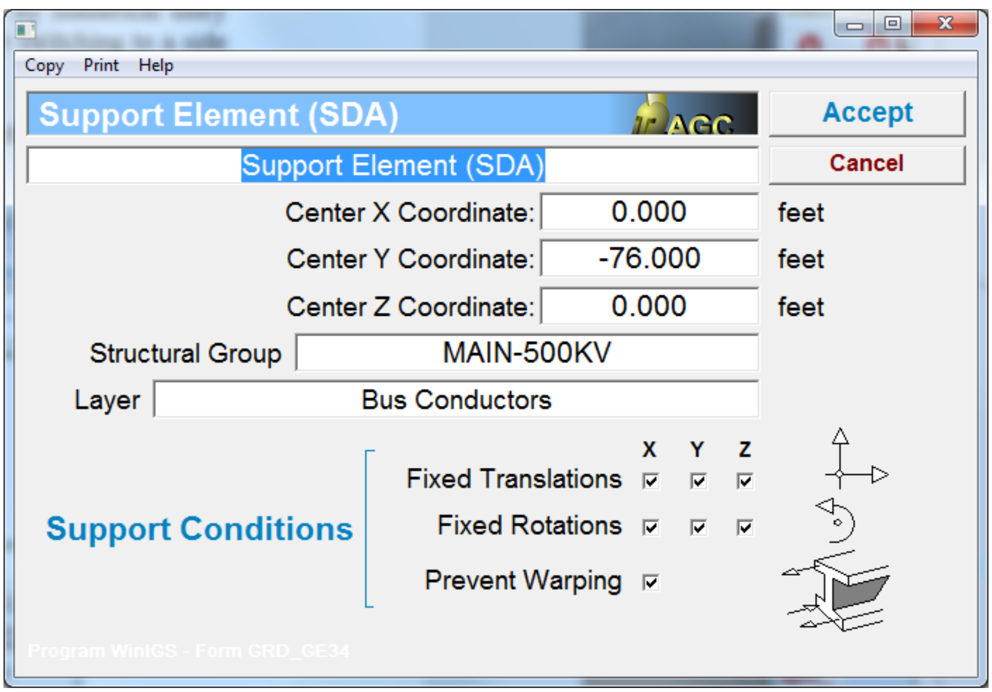


Figure 1.4b: Support Element Parameters Dialog Window

1.5 The SDA Mechanical Connector

The SDA mechanical connector element connects beam elements with several connector types, namely: springs, sliders, hinges, ball and universal joints. Figure 1.5a shows an example of a radial slider connector. This connector allows sliding along the horizontal beam axis.

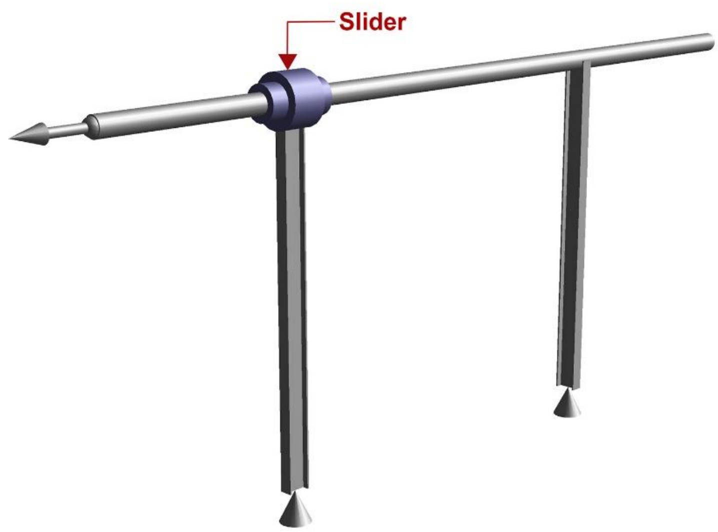


Figure 1.5a: Slider Example – 3D Rendered View

connector elements are added using the mouse in top-view mode (x-y) using a single left-click at the desired location. The z- coordinate is initially set to zero, and can be later modified either by numerical entry (using the properties window) or graphically (using the mouse). The connector element has two nodes (See Figure 1.5b) – the main node and the control node.

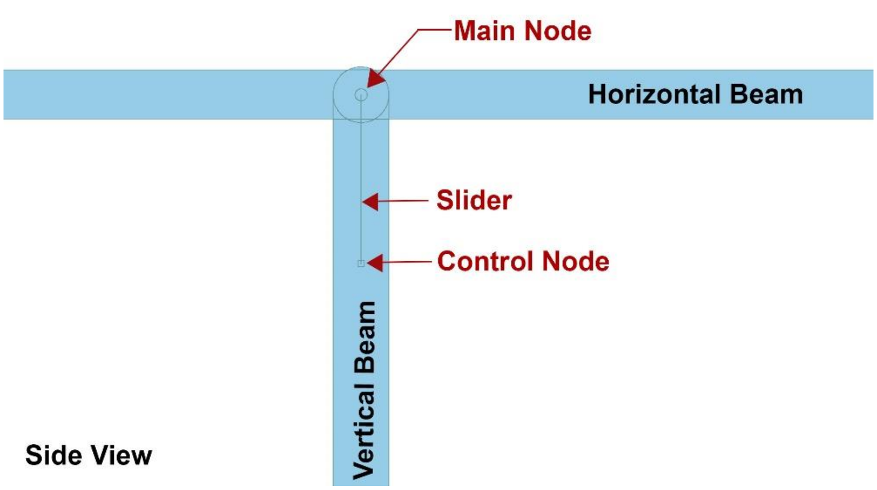


Figure 1.5b: Slider Example – Side View

The main node defines the actual position of the connector, while the control node determines the orientation of the slider. In this example a “Radial Slider” option has been selected, and the control node defines the direction of the slider radial direction.

The connector properties window (opened by a left double click on the connector element) is illustrated in See Figure 1.5c. The connector type is selected via the 8 radio buttons located on the left side of the window. The orientation of the slider is set by the location of the control node (or beam identifier) and can be also rotated about the axis defined by the main and control nodes by user selected angle (Orientation Field) specified in degrees.

The spring constants field is only applicable if the connector type is set to “Spring”, and is otherwise ignored.

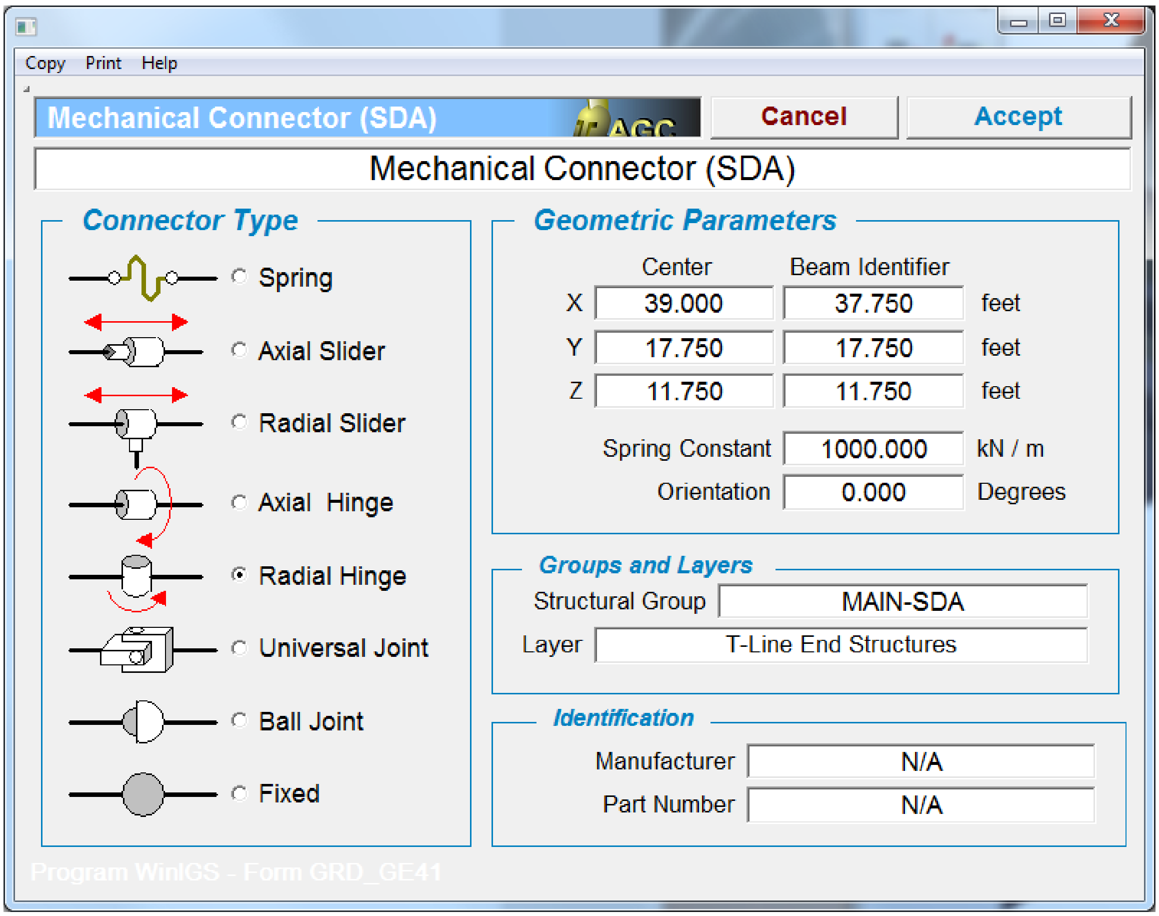


Figure 1.5c: Mechanical Connector Element Parameters Dialog Window

1.6 The SDA Mechanical Source

The SDA mechanical source element applies a user specified force or moment at a selected point of a structure. Figure 1.6a shows an example of a mechanical force element applying a horizontal force at the end point of a horizontal beam.

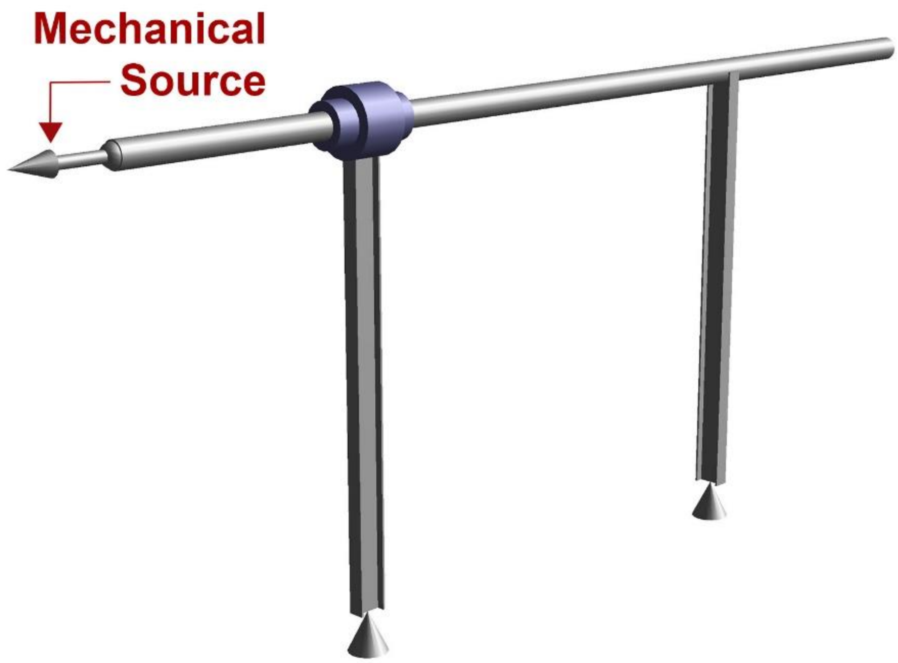


Figure 1.6a: Mechanical Source Example – 3D Rendered View

The mechanical source properties dialog window (opened by a left double click on the source element) is illustrated in Figure 1.6b. The source waveform is a sinusoidal of a user selected amplitude, frequency, and phase angle, namely:

$$F = a\cos(2\pi f + b)$$

where a is the amplitude, f is the frequency and b is the phase angle. Note that setting the frequency *f* and the phase angle *b* to zero results in a constant force or moment of amplitude *a*.

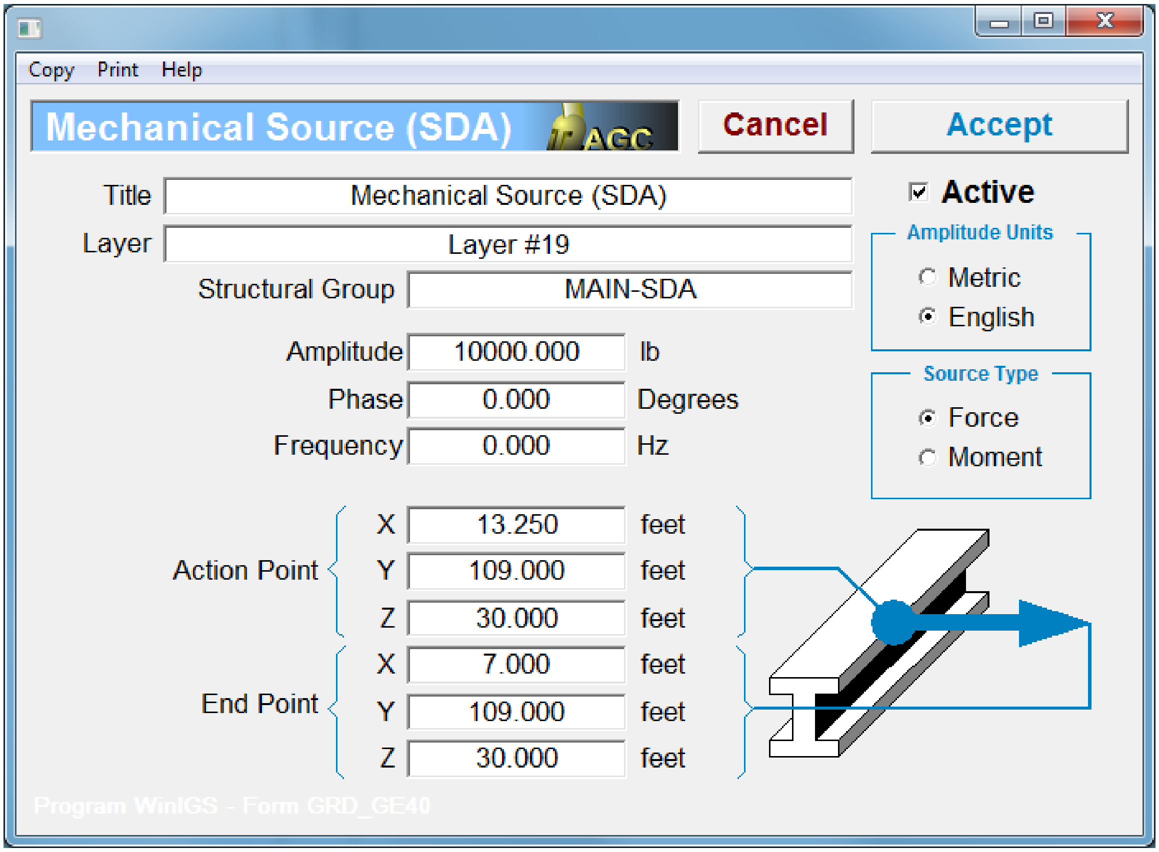


Figure 1.6b: Mechanical Source Element Parameters Dialog Window

1.7 The SDA Structural Dynamics Meter

The SDA Structural Dynamics Meter element monitors a quantity computed during the dynamic simulation such as a displacements, forces, moments etc. at a user selected location. monitored quantities are available for plotting. Some of these quantities can be compared to maximum permitted values. Monitored quantities exceeding permitted values can be listed in tabular reports. Figure 1.7a illustrates the representation of an example meter element within the 3D editor. Note that a meter element location is determined by two nodes: the main node, where the computed quantity is monitored, and optionally, an element identifier node. The element identifier node resolves the ambiguity of which quantity is monitored in cases where the main node is located at an intersection of two or more beams. Consider for example that the meter in Figure 1.7a is set to monitor the shear force at point the intersection of the vertical and horizontal beam. Placing the element identifier on a point along the vertical beam, sets the meter to monitor the shear force on the vertical beam.

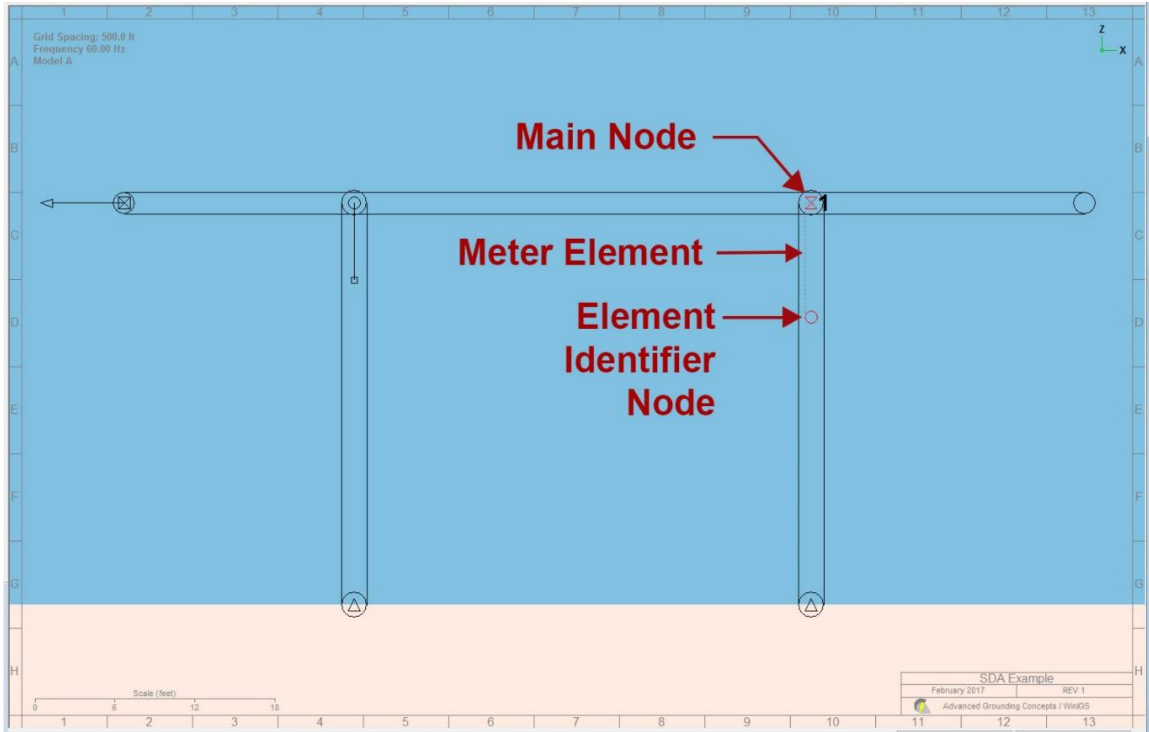
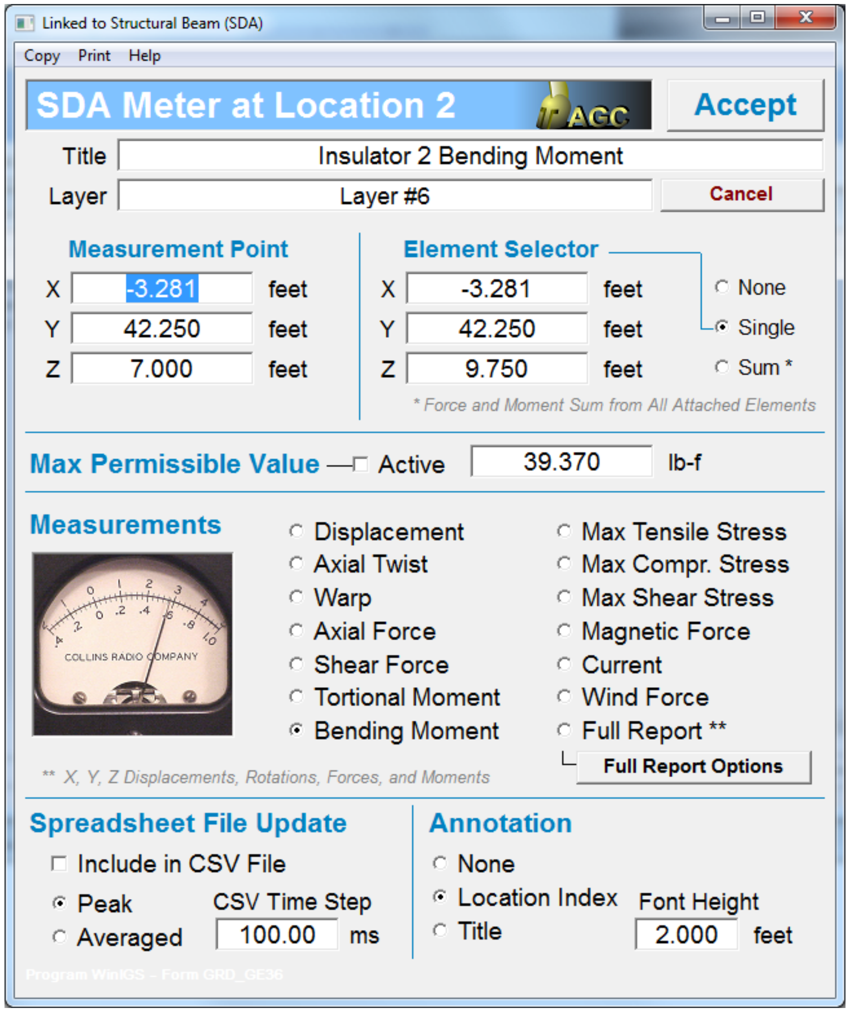


Figure 1.7a: Meter Element Example

The meter element source properties dialog window (opened by a left double click on the meter element) is illustrated in Figure 1.7b. The monitored quantity is selected by activating one of the 14 radio buttons in the Measurements control group. All selections except the ***Full Report*** option selects a single quantity. The full report option allows selection of multiple quantities. The Full Report Options button opens a second dialog window which allows selection of any number of displacements, forces, rotations and moments.



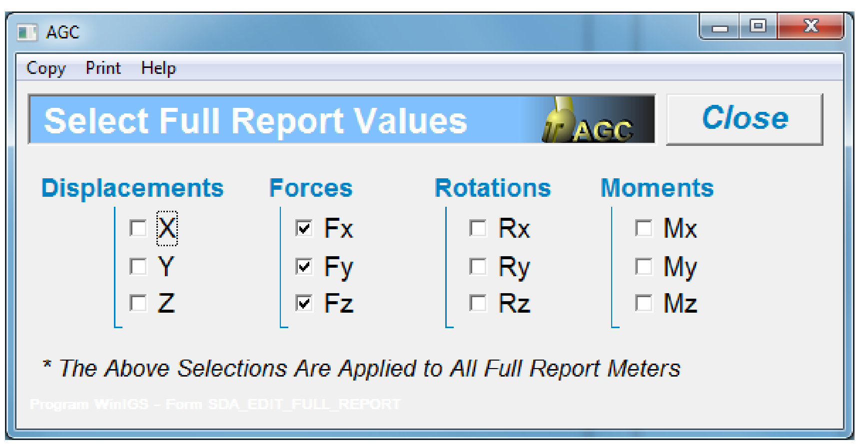


Figure 1.7b: Meter Element Parameters Dialog Windows

The selected measured quantity is retrieved from the beam coincident to the meter "Measurement Point" location. If more than one beams are attached to the meter Measurement Point, a second point (called the meter "Element Selector") is used to uniquely identify the beam of interest. To activate the Element Selector point set the Element selector radio button to "**Single**", an place this point along the length of the desired beam. (See Example in Figure 1.7a)

Note that if the Element selector radio button is set to "**Sum**", the measurement quantity is derived from the vector sum of the quantities of all the beams attached to the Meter Measurement Point. This feature can be used to measure the total forces and moments that are acting on a foundation point. An example of such an application is shown in Figure 1.7c.

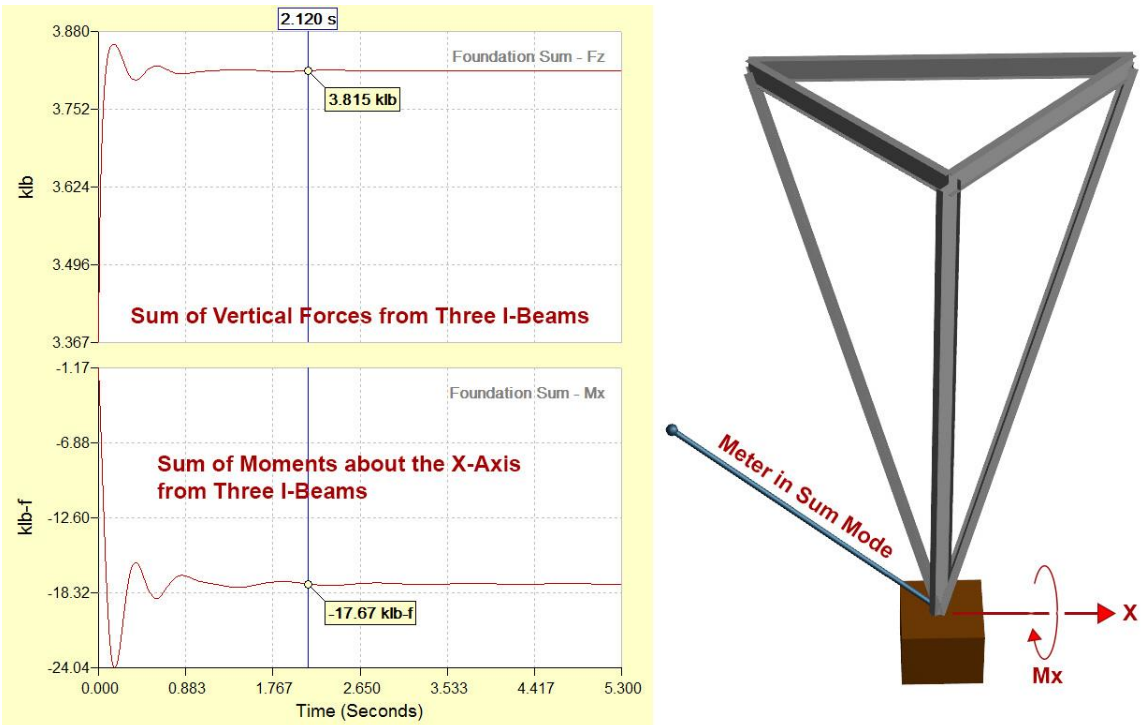


Figure 1.7c: Application Example of Meter Measuring Forces and Moments at the foundation center-point in *Sum Mode*

3. Simple Substation Rigid Bus

3.1 Introduction

This section presents the structural dynamic analysis of a simple three phase rigid bus structure. The analysis simulates the mechanical response of the rigid bus during an electrical fault. Specifically, the program computes the displacements and stresses occurring on the buswork components due to the magnetic forces generated during the electrical fault. The WinIGS data files for the example system are provided under the study case name: IGS_SDA_TGUIDE_CH03.

This example closely follows the IEEE-605 Annex F example, so the results can be compared with the results found in the standard. The rigid bus structure is illustrated in Figure 3.1 (reproduced from page 177 of the IEEE Std-605)

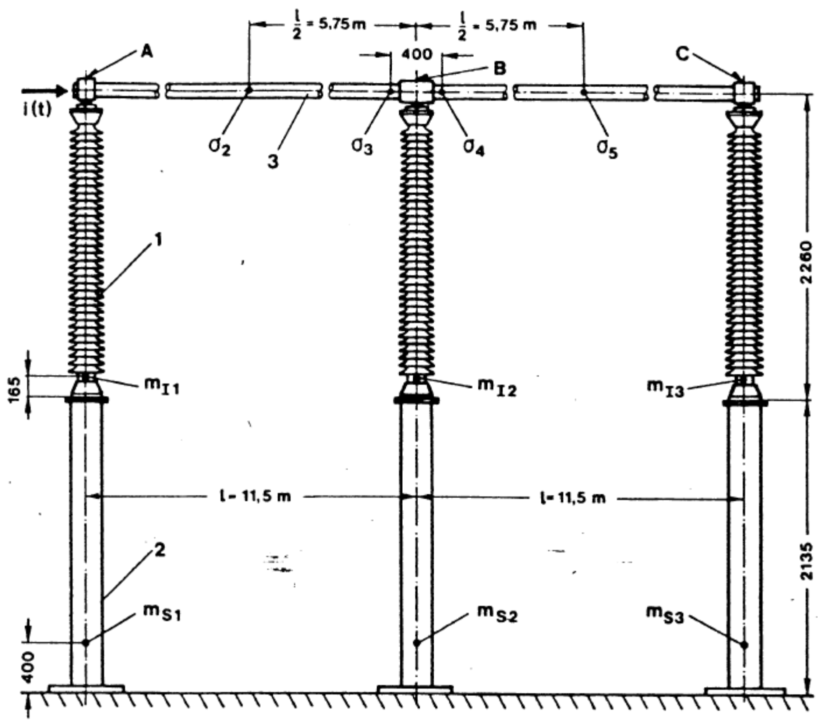


Figure 3.1: Side view of Rigid Bus Example
(From IEEE Std-605 Annex F, page 177)

3.2 Model Description

The WinIGS model for this example is provided under file name IGS_SDA_TGUIDE_CH03. Use command **Open** of the **File** menu or click on the icon:

 to open this study case. Once the study case is opened, the network editor window appears showing the system single line diagram, illustrated in Figure 3.2. The system network model includes a 3-phase source feeding the rigid bus model at the node labeled BUS2. A load three connectors at BUS3. The rigid bus model is connected between buses BUS2 and BUS3. A phase to phase (A-B) fault is applied at the load side bus, resulting in 15.6 kA of fault current flowing through the rigid bus model.

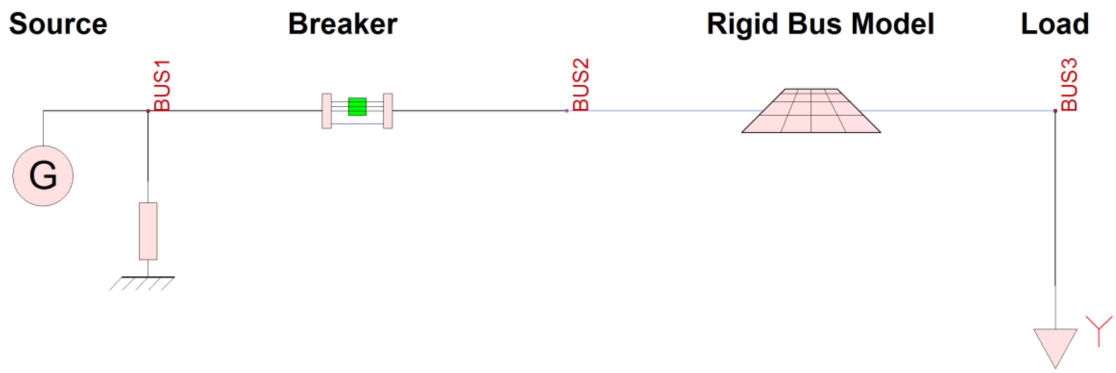


Figure 3.2: Network Model Single Line Diagram

Note that the parameters of the source (illustrated in Figure 3.3) have been selected to match the Phase-to-Phase short circuit current specified in Annex F Std-605 example. Specifically, assuming equal positive and negative sequence source impedances, the following relationship between Line-to-Line fault current, short circuit capacity and nominal L-L voltage holds:

$$I_f = \frac{S}{2 \times V}$$

thus:

$$I_f = \frac{6863.9MVA}{2 \times 220.0kV} = 15.6kA$$

Figure 3.4 shows the rigid bus model in wireframe and rendered views. The bus consist of three aluminum pipes suspended on porcelain post insulators mounted on steel beams. As per Annex F example the total length of the bus is 23 meters, and the distance between the phases is 1 meter. The insulator height is 2.3 meters, and the insulator support beams are 2.1 meters.

The section properties of the bus pipes, insulators, and support steel beams are given in Table 3.1 (reproduced form page 177 of the IEEE Std-605). Sections for the bus aluminum, pipes, insulators and support steel beams were selected from the libraries to match the Annex F example data in Table 1. Note that since the detailed construction parameters were not available some assumptions had to be made. For example a square section was assumed for the support beams.

Table 3.1: Section Properties (form Annex F, IEEE Std-605)

Characteristic	Conductor	Insulator	Support
Length *L*	span of 11.5 m (including portion in conductor clamps)	2.10 m	2.135 m
Material	Aluminum	Porcelain	Steel
Young's modulus *E*	70 GPa	30.6 GPa	206 GPa
Area *A*	2.238E-3 m^2	36.14E-3 m^2	4.714E-3 m^2
Moment of inertia *I*	3.704E-6 m^4	76.08E-6 m^4	26.83E-6 m^4
Mass	6.04 kg/m	180 kg	36.8 kg/m

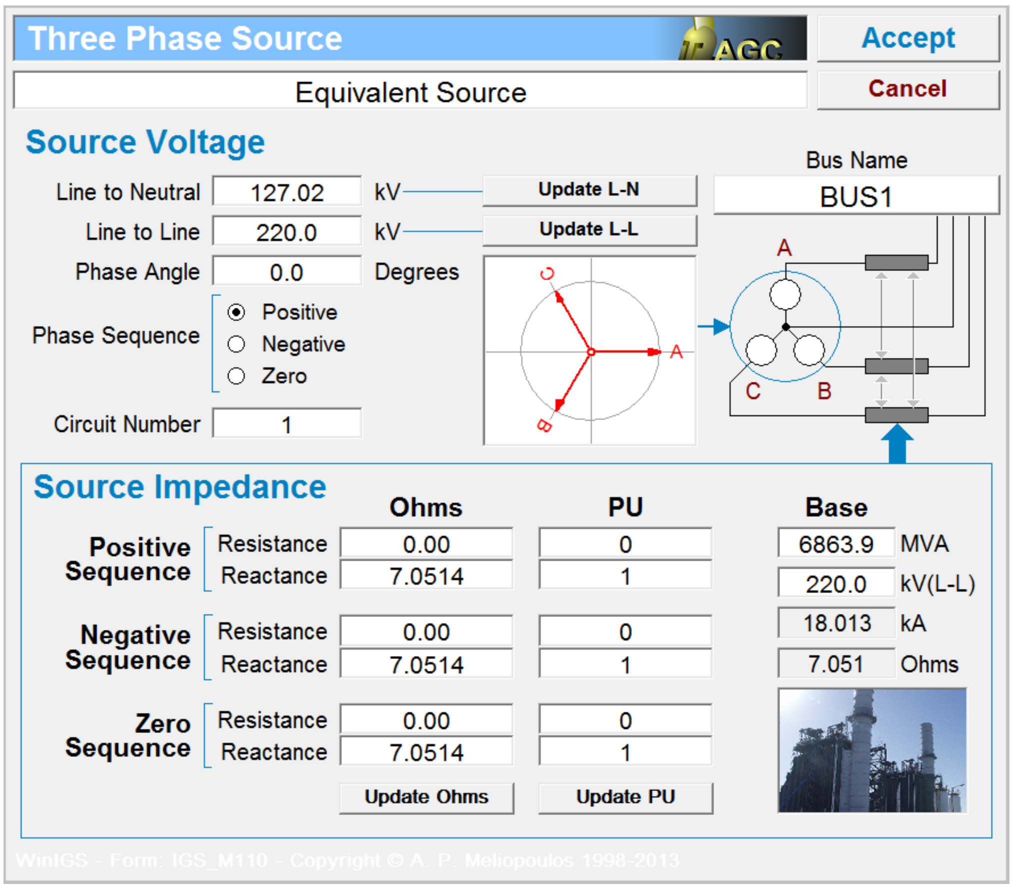


Figure 3.3: Source Parameters

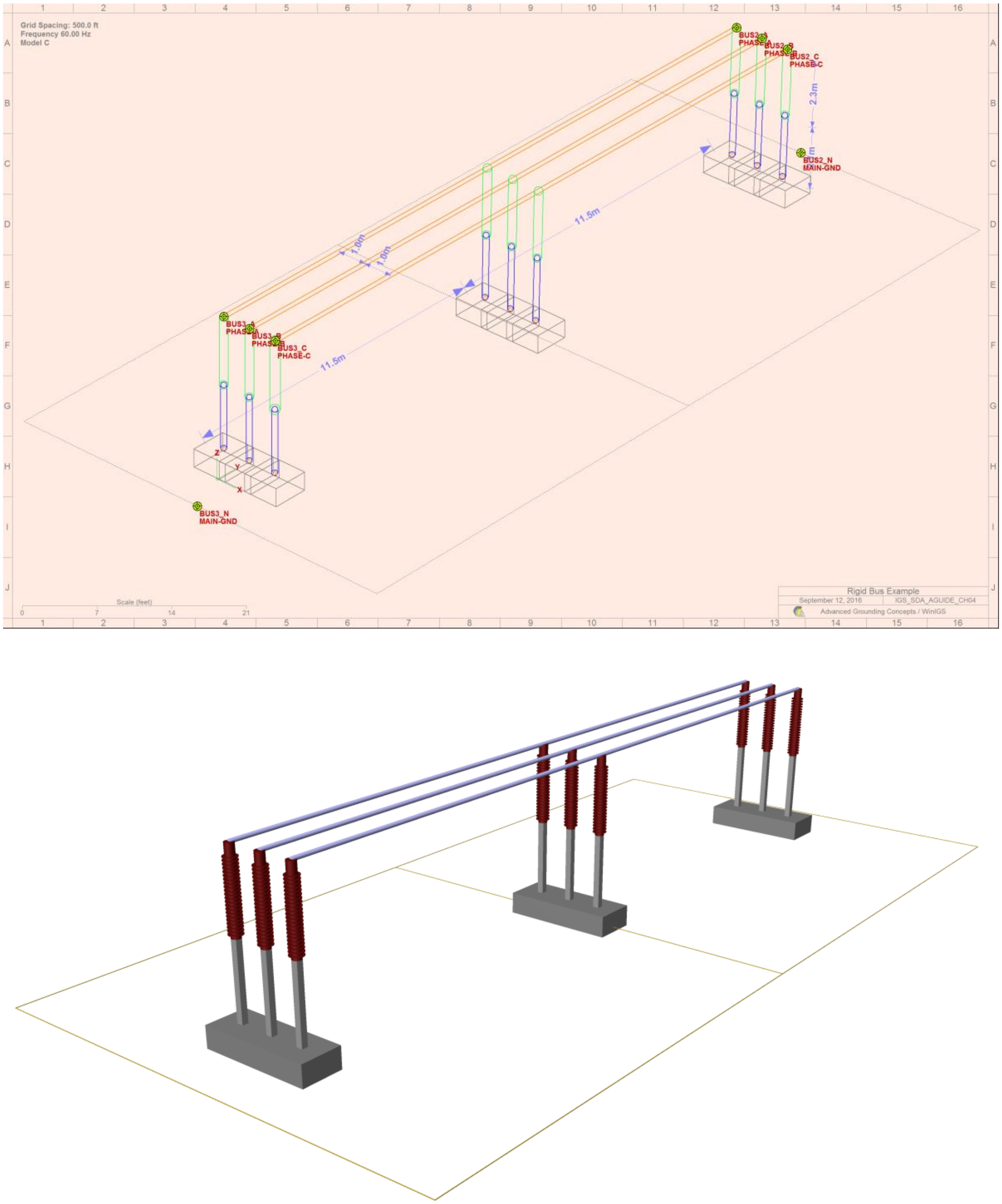


Figure 3.4: Rigid Bus Model Single Wireframe and Rendered Views

3.2 Analysis

The analysis begins by executing a Line-to-Line fault solution which computes the electrical currents flowing through the rigid bus under consideration. Click on the **Analysis** button, select **Fault Analysis**, and click on the **Run** button (See Figure 3.3).

Next on the fault definition dialog window, select **Fault at a Bus**, select **BUS 3**, **Line to Line** radio button, and faulted phases **A** and **B**. Finally click on **Execute** button to perform the fault analysis calculations.

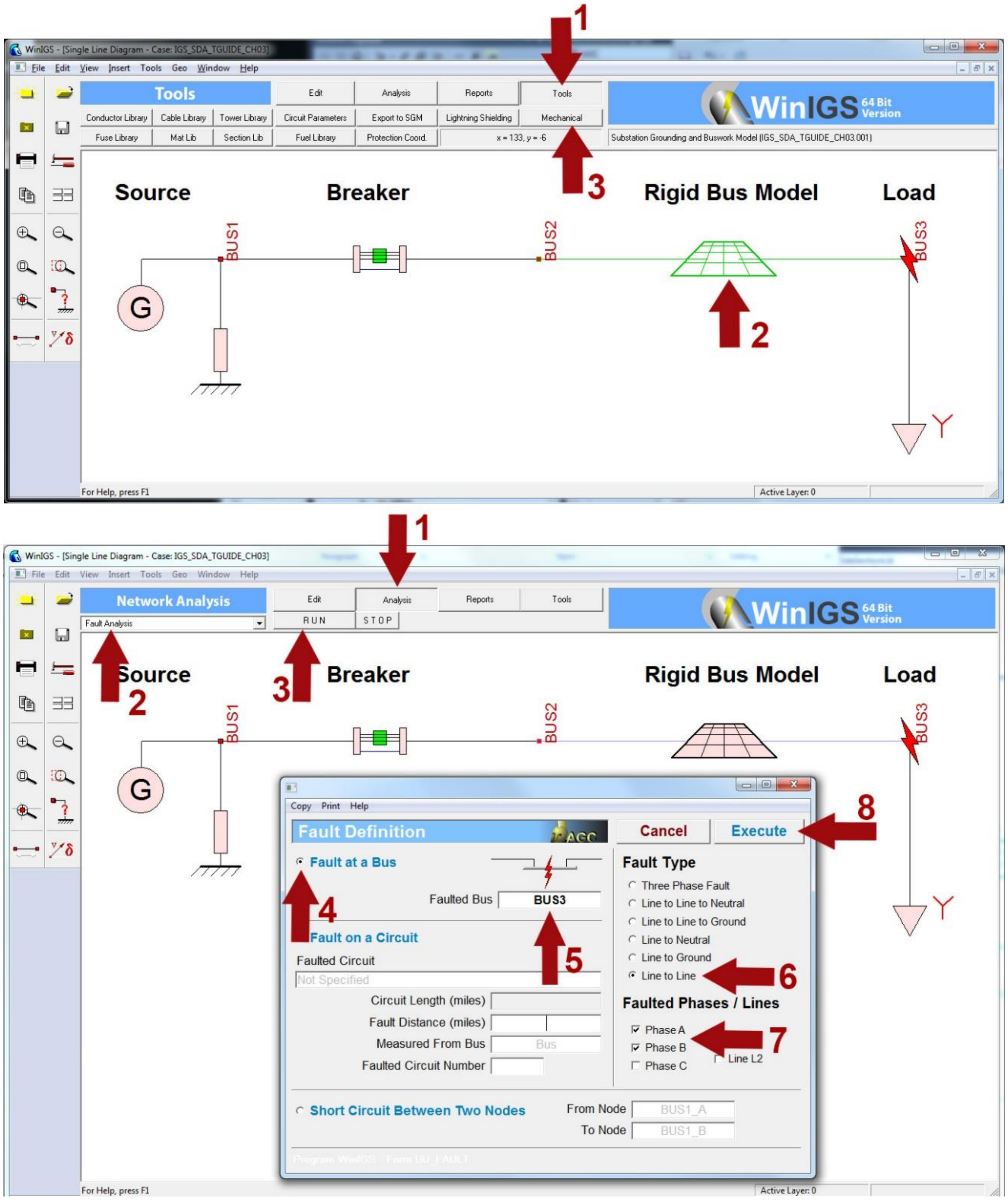


Figure 3.5: Fault Analysis Steps

Next switch to **Tools** mode, select the rigid bus model object and click on the Mechanical button (See Figure 3.4). Next click on the Start button to perform the bus dynamic simulation (See Figure 3.7)

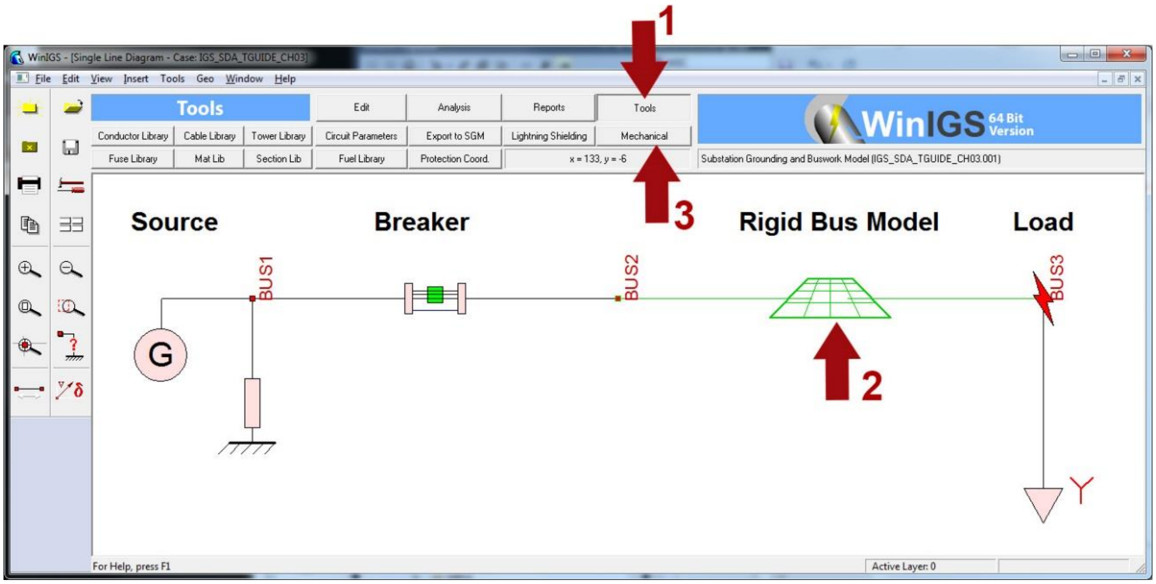


Figure 3.6: Switching to SDA mode

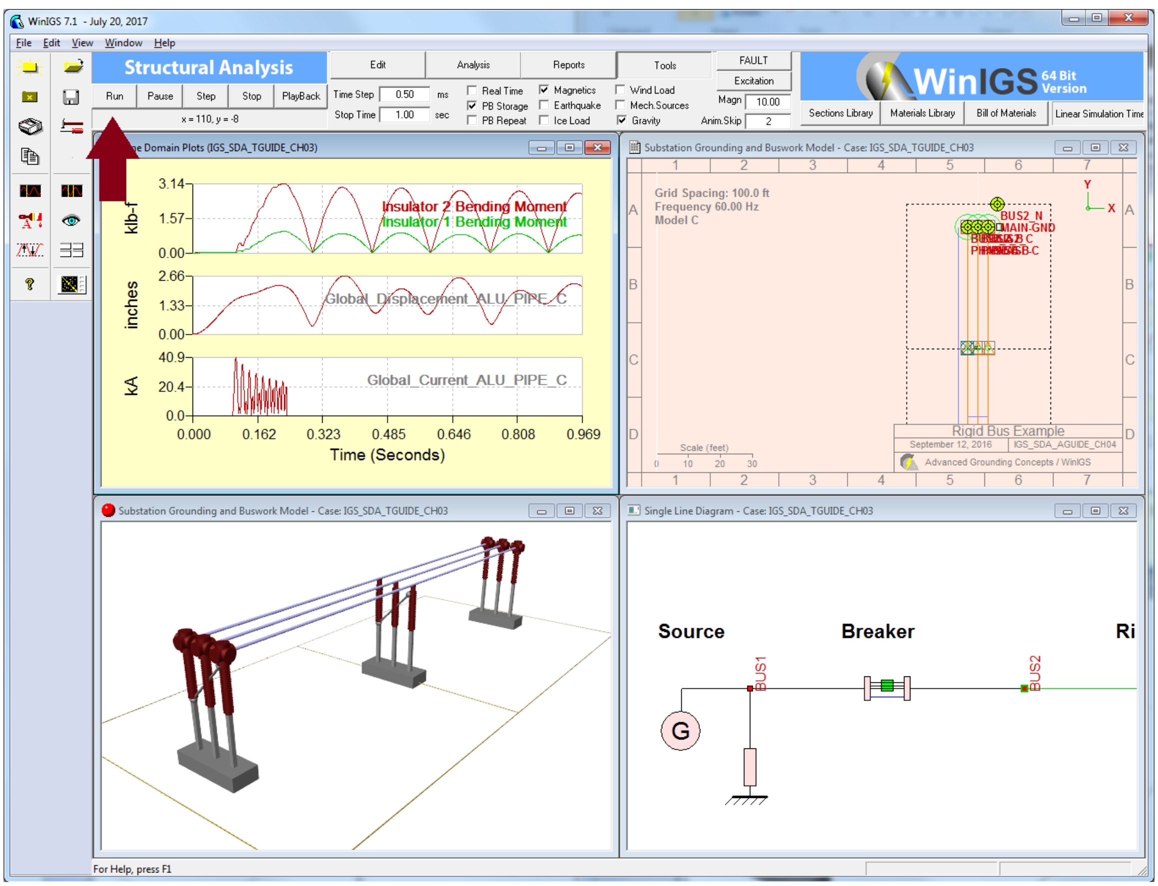


Figure 3.7: Executing SDA Simulation

3.3 Comparison with IEEE-Std605 Standard Formulas

Two bending moment meters were added at the base of insulators 1 and 2 (Points MI1 and MI2 in the IEEE Std-605 Annex F example). The computed bending moments are shown in Figure 3.8 plotted versus time. The plot includes the maximum displacement and the electric current. All plots are absolute values. Note that the maximum bending moments reach about 1.0 and 3.3 kNm for insulators 1 and 2 respectively. Also, the maximum displacement is 44 millimeters. These results are within reasonable agreement to the results given in IEEE Std-605 Annex F, which are reproduced in Figure 3.9.

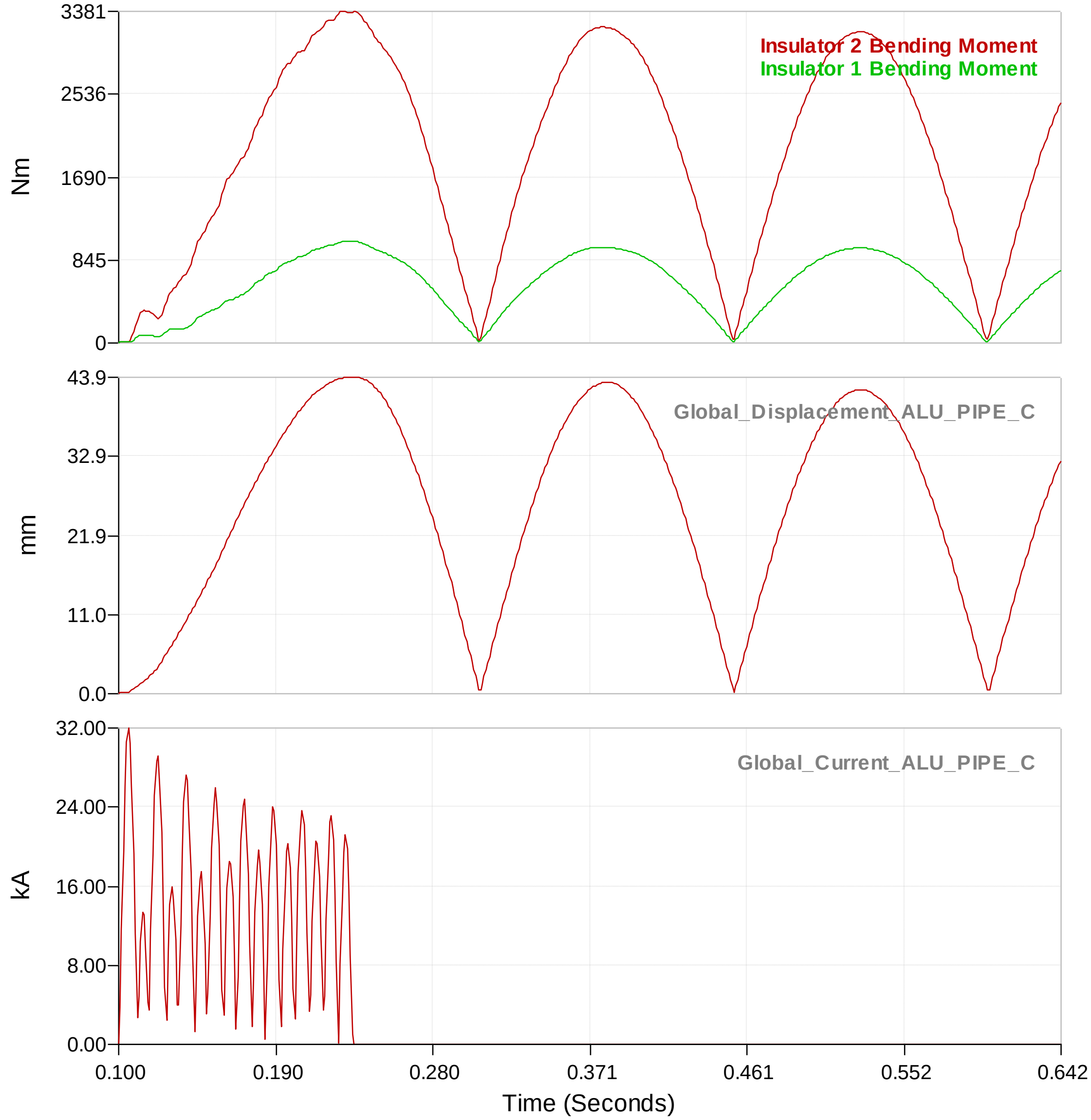
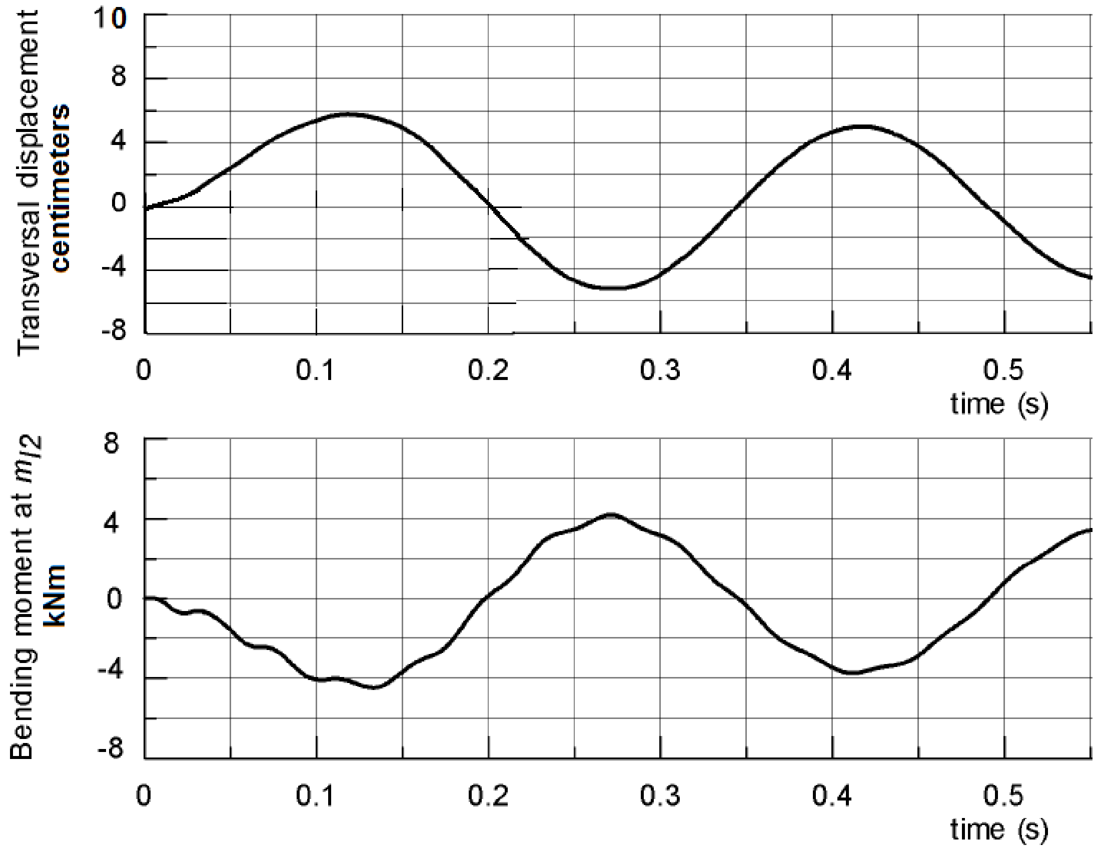


Figure 3.8: Plot of WinIGS-SDA Simulation Results



(a)

Result		Experimental	Finite element
M_{I1}	Value (N-m)	1600	1335
	% error/exp.	—	17%
M_{I2}	Value (N-m)	5080	4467
	% error/exp.	—	12%

(b)

Figure 3.9: Results Reproduced from IEEE Std-605 Annex F
(a) Plot of displacement and bending moment versus time
(b) Bending Moment Maximum Values

4. Simple Substation Strain Bus

4.1 Introduction

This section presents the structural dynamic analysis of a three phase strain bus structure. The analysis simulates the mechanical response of the strain bus during an electrical fault. Specifically, the program computes the displacements and stresses occurring on the buswork components due to the magnetic forces generated during the electrical fault. The WinIGS data files for the example system are provided under the study case name: IGS_SDA_AGUIDE_CH04.

4.2 Model Description

Open the study case titled: IGS_SDA_AGUIDE_CH04. Use command **Open** of the **File** menu or click on the icon:  to open the existing study case data files. Once the study case is opened, the network editor window appears showing the system single line diagram, illustrated in Figure 4.1. The system network model includes 4 transmission lines, four equivalent sources connected at the remote end of each line and an equivalent load model. The stain bus under study is connected between busses BUS1 and LOAD.

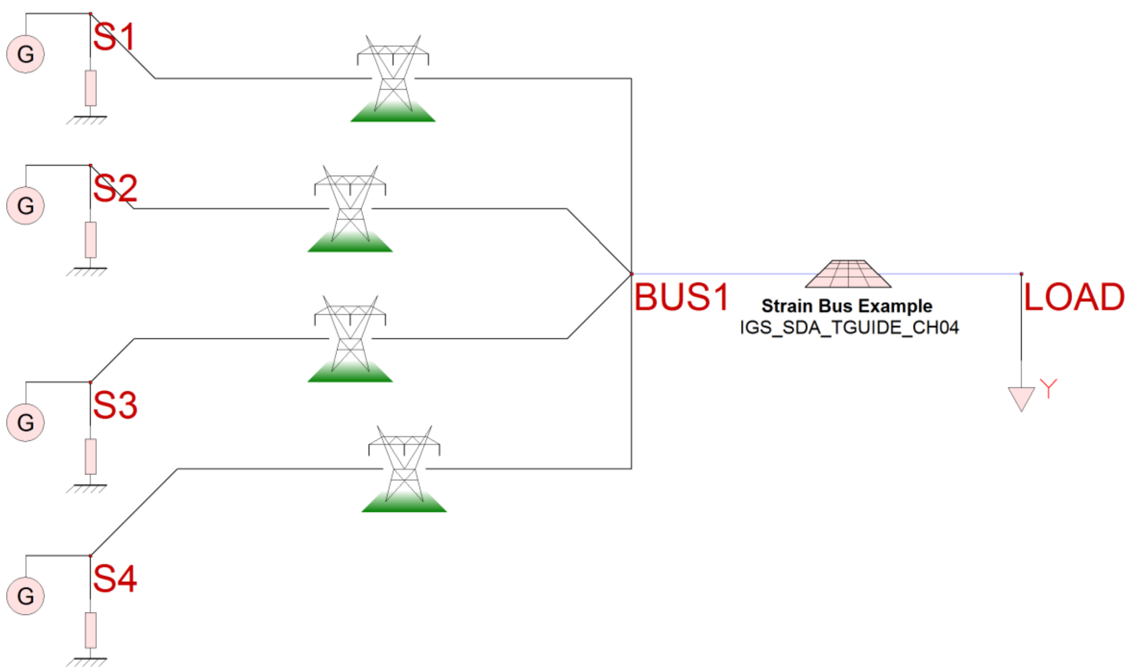


Figure 4.1 Single Line Diagram of Example System IGS_SDA_AGUIDE_CH04

Double click on each source element to examine the source parameters (See also Figure 4.2).

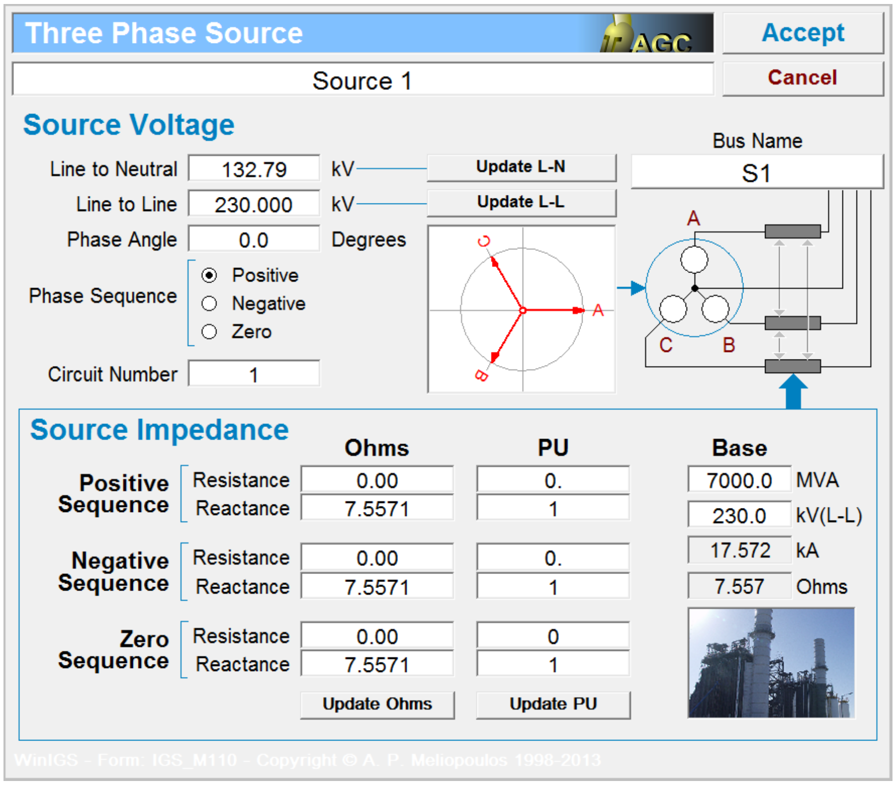
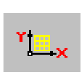
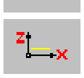


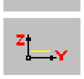
Figure 4.2 Source 1 Parameters (All Sources are Identical)

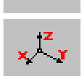
In order to inspect the strain bus model of this example, open the grounding system editor window by double clicking on the grounding model icon:

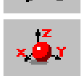
The grounding system editor provides a graphical CAD environment with extensive display and editing capabilities. The grounding system along with the modeled buswork can be displayed in top view, side view, or perspective view. Use the following toolbar buttons to switch among these viewing modes, as follows:

1  Top view (See Figure 4.3)

2  Side View

3  Side View

4  Perspective View (See Figure 4.4)

5  Rendered Perspective View (see Figure 4.5)

By default the top view of the modeled system is shown. At any view mode you can zoom using the mouse wheel and pan by moving the mouse while holding down the mouse right button. In the perspective view mode, you can also rotate the view point by holding down both the keyboard Shift key and the right mouse button.

The modeled system consists includes a 3-phase strain bus (shown in blue in the top view) and support structures (dashed lines). Note that in this example the support structures are represented by “Non-SDA” elements. Non-SDA elements are ignored by the structural analysis solver.

Figure 4.8 illustrates the bus support structure detail. The flexible conductors are connected to insulators, which are connected to SDA support elements. A support element (triangle symbol) at the end of each insulator represents the attachment of the insulators to the horizontal beam of the support structure. Figures 4.9 through 4.12 illustrate the parameters of the major elements.

In order to compute the magnetic forces that develop on the bus conductors during electrical faults, the electric current through the bus conductors must be evaluated first. For this purpose the model includes a physical representation of the conductive paths along each phase of the bus and the connections to the external network. Note that the connections of the bus conductor ends to the external network components (transmission lines, transformer etc) are made via eight interface nodes, represented by the symbol , with labels based on the corresponding bus names to which they are connected. Each of these interface nodes produces a connecting terminal on the network view (single line diagram) which are attached to the appropriate devices. The connections at each terminal node of the single line diagram can be inspected by double clicking on the node. Figure 4.13 illustrates the inspection window for the connections between the modeled buswork and the substation transformer. This window is opened by double clicking on the BUS5 node.

Figure 4.3

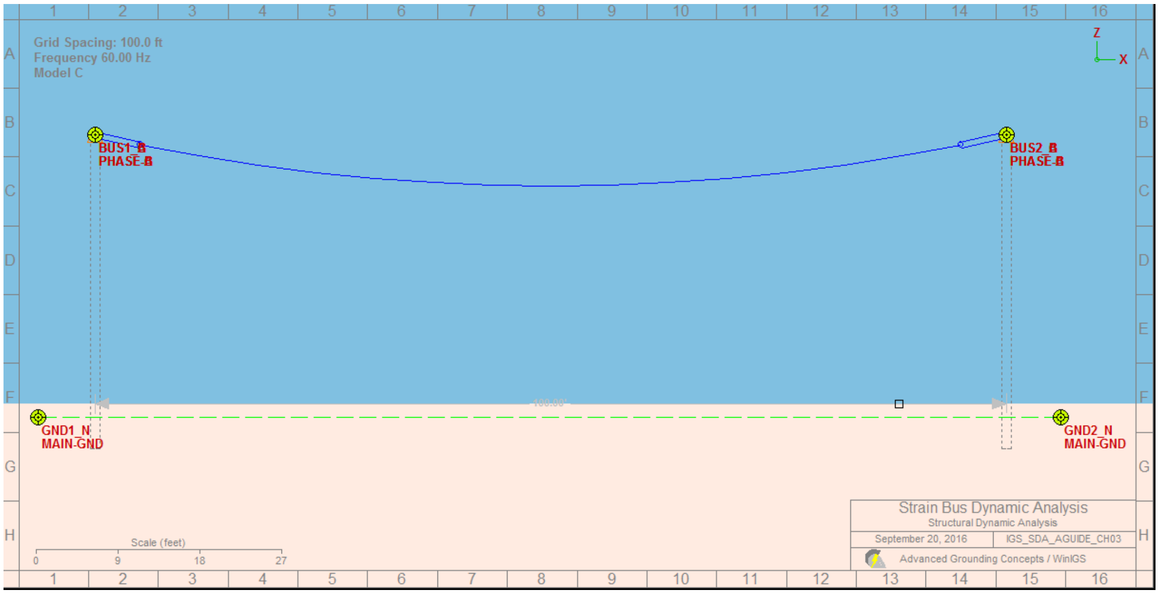


Figure 4.6: 3-Phase Rigid Bus Model – Perspective View

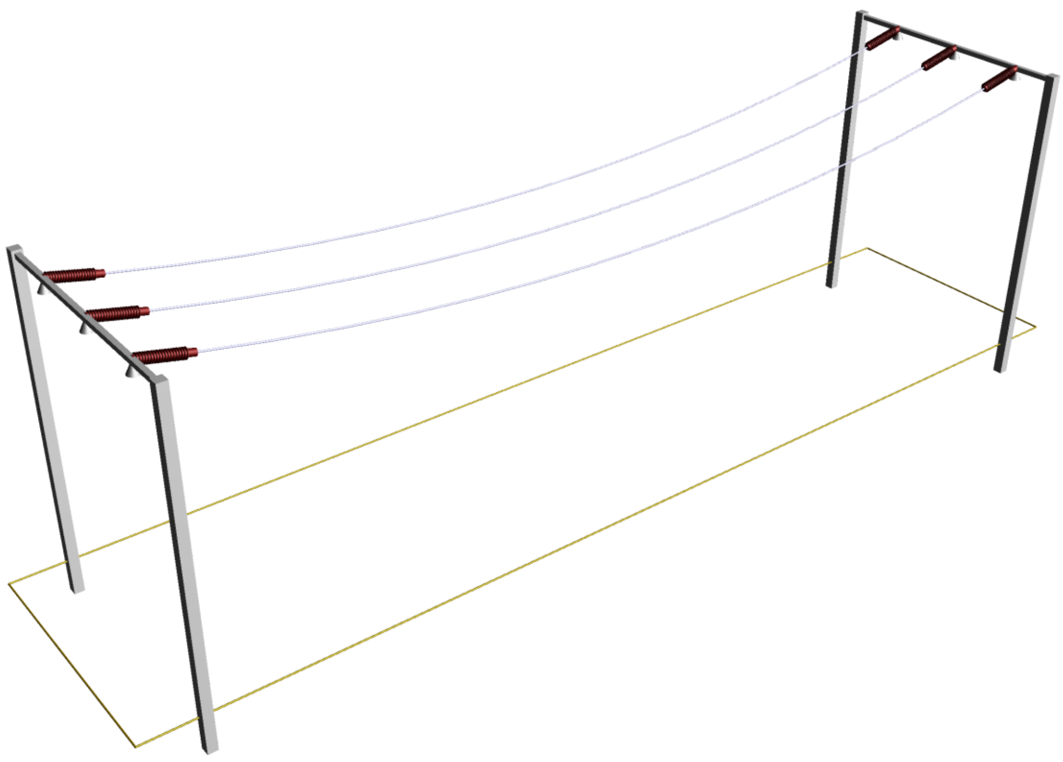


Figure 4.7: 3-Phase Strain Bus Model – Rendered Perspective View

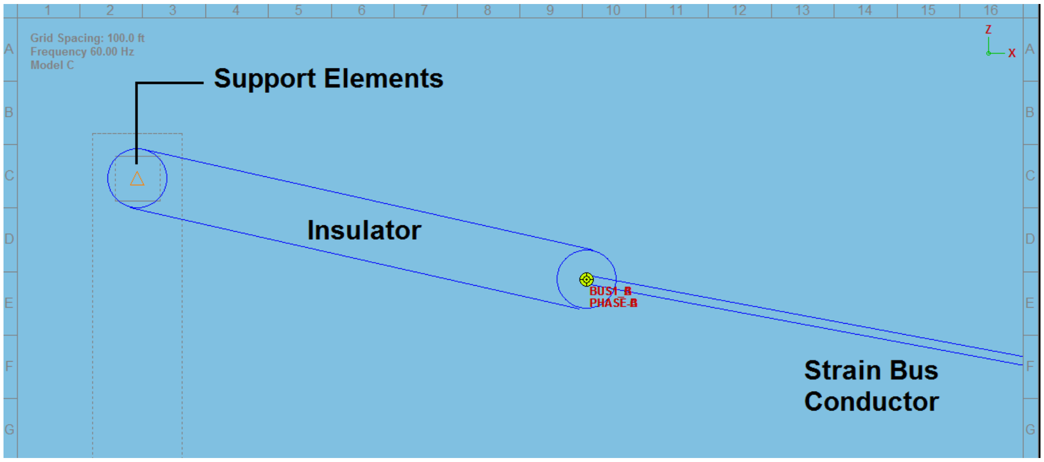


Figure 4.8: Bus Model – Side View Detail

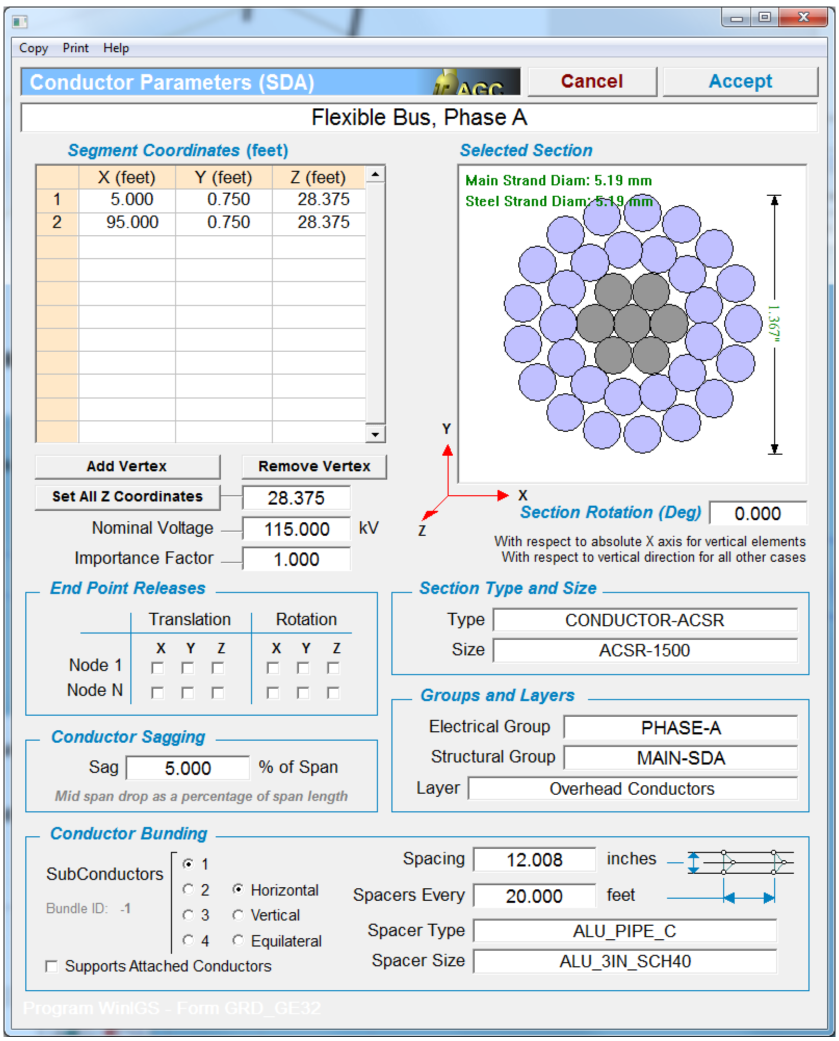


Figure 4.9: Strain Bus Phase-A Conductor Parameter Form

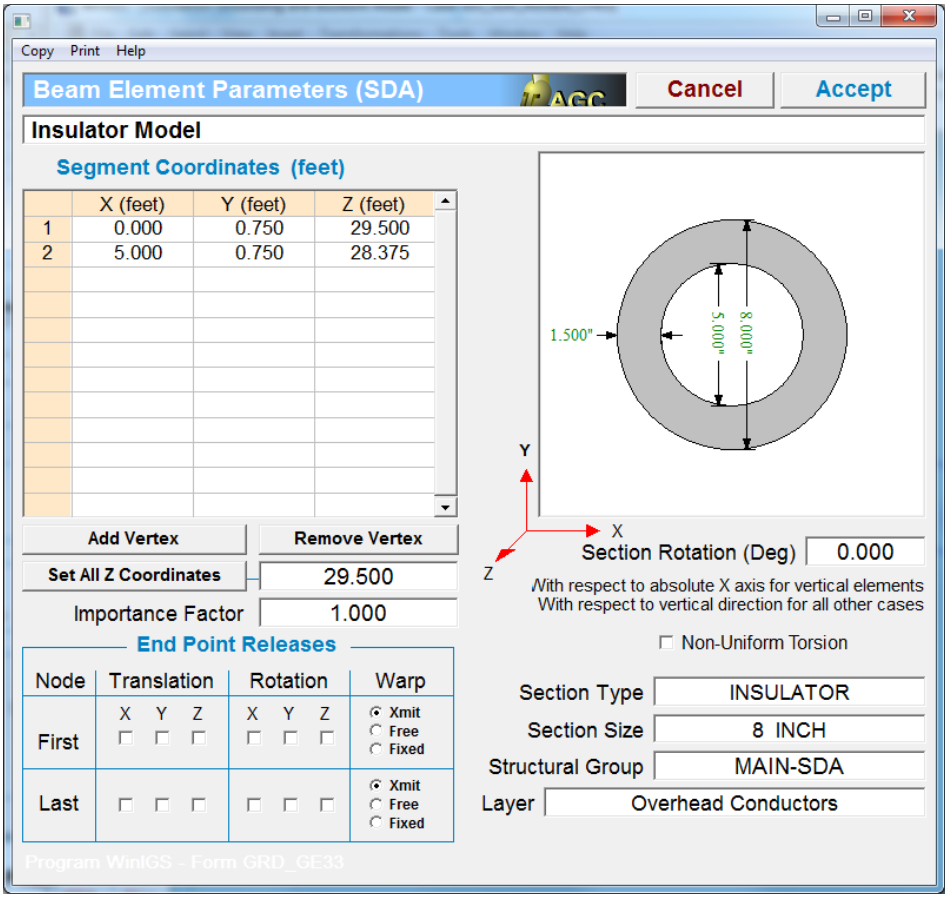


Figure 4.10: Insulator Model Parameter Form

4.3 Electrical Fault Analysis

The fault analysis of this system comprises of two steps.

- Step 1 – Equivalent Impedance Analysis
- Step 2 – Electrical Fault Analysis

The fault analysis step can be performed using any if the fault analysis options provided by WinIGS. In this example a three phase fault on the 500 kV bus (BUS5) is selected. The electrical fault analysis results are automatically picked up by the structural analysis solver in order to compute the resulting magnetic forces on the modeled buswork.

In addition to the fault current levels, the X/R ratio at the fault location is required in order to estimate the DC offset of the current waveforms. The X/R ration is obtained by performing Port Thevenin Equivalent analysis at the fault location, as follows:

- Click on the **Analysis** button
- From the pull-down list control select the "**Port Thevenin Equivalent**" option

- Click on the **Run** button to open the equivalent impedance analysis form, illustrated in Figure 4.14.
- Select the **3–Phase Bus Port** radio button, and the **Positive Sequence** option.
- Click on the **Execute** button, and note the reported X/R ratio.

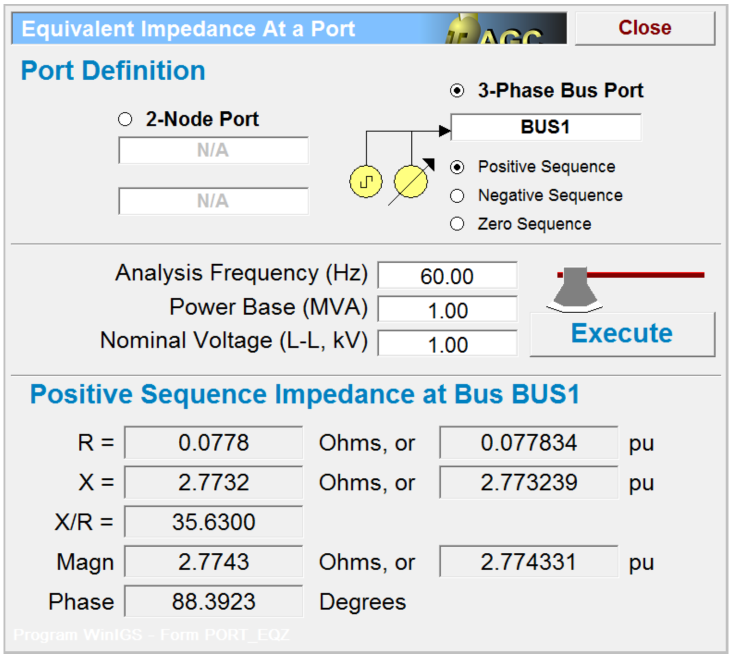


Figure 4.14: Equivalent Impedance Report

The procedure for performing fault analysis is as follows:

- Click on the **Analysis** button
- From the pull-down list control select the "**Fault Analysis**" option
- Click on the **Run** button to open the fault analysis option form.

Note that all these controls are located along the top side of the main program window frame.

The fault analysis option form is illustrated in Figure 4.15. Ensure that the selected option are as illustrated in this Figure. Specifically, select **Fault at a Bus** radio button, Set **Fault Type** to **Three Phase Fault,** select **Faulted Phases / Lines** to be **Phase A, Phase B,** and **Phase C**, and then click on the **Execute** button.

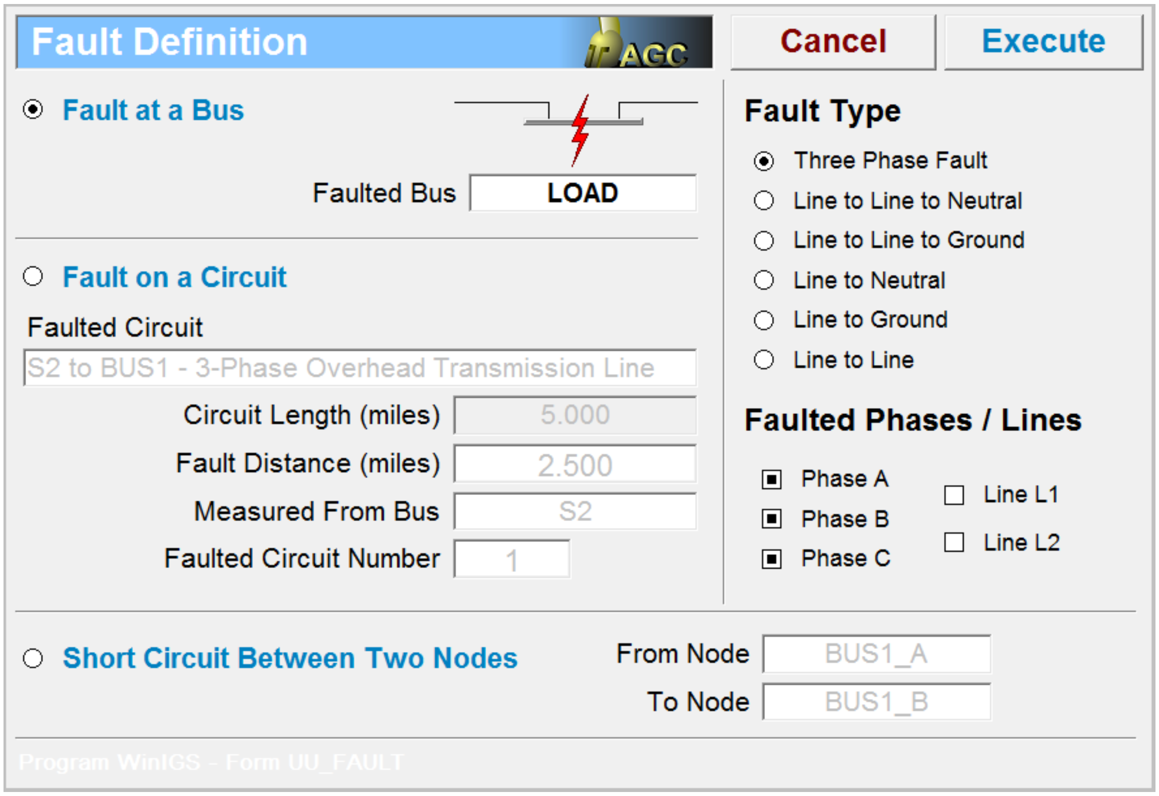


Figure 4.15: Fault Analysis Options

Once the analysis is completed, a pop-up window appears indicating the completion of the analysis. This window, illustrated in Figure 4.16, also displays the computed fault current. Click on the **Close** button to close this window, and then click on the **Reports** button to enter into the report viewing mode.

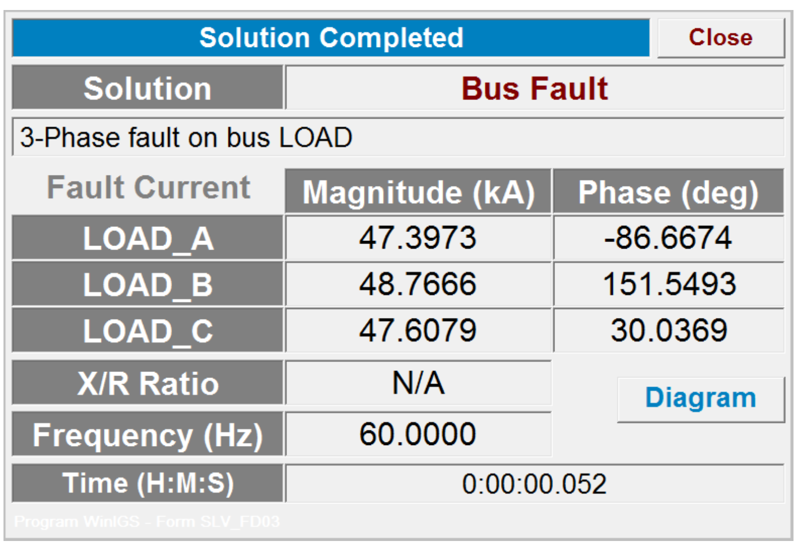


Figure 4.16: Fault Analysis Completion Report

While in Reports mode, a set of "*radio buttons*" appears along the top of the main program window frame, which allows selection of the report type. From these buttons, select the **Graphical I/O report**, and then double click on the grounding system icon to view the Device Voltage and Current Report. This report is illustrated in Figure 4.17. Note that the current circulating through the modeled bus is about 62 kA.

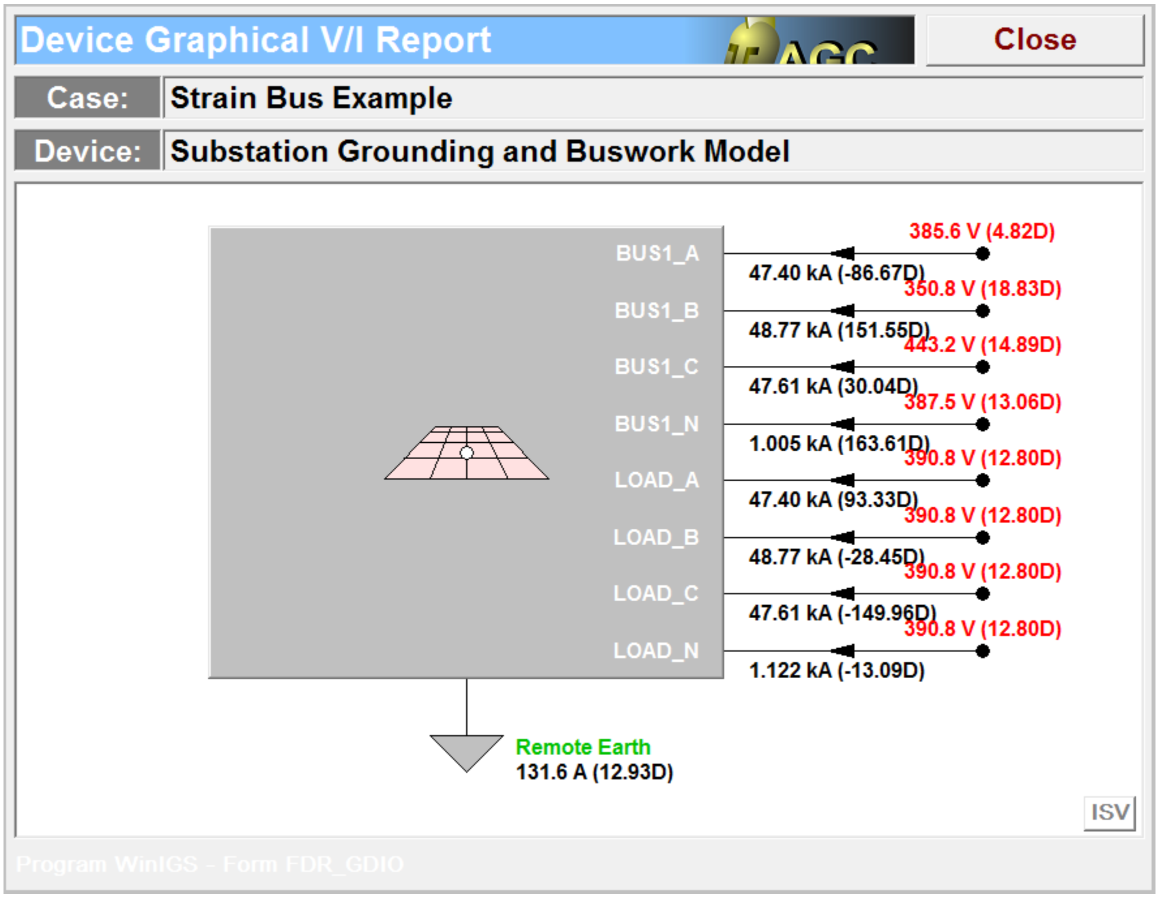


Figure 4.17: Device Voltage and Current Report

4.5 Structural Dynamic Analysis (SDA)

The structural dynamic analysis is performed in three steps:

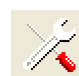
Step 1: Perform analysis with gravity excitation only. The objective of this step is to compute and store the appropriate steady state conditions. In order to speed up the decay of the oscillations, it is recommended that the system damping ratio is set to a high value (e.g. 1.0).

Step 2: Perform analysis with both gravity and magnetic force excitation. The objective of this step is to identify the locations where maximum stresses occur during the simulated electrical fault. The initial conditions are obtained from the results of Step 1. The system damping ratio is set to a typical value for rigid bus structures (e.g. 0.1).

Step 3: Repeat analysis with both gravity and magnetic force excitation. The objective of this step is to generate plots of stress versus time at the points of maximum stress identified in Step 2. The initial conditions are again obtained from the results of Step 1.

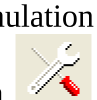
In order to access the Structural Dynamic Analysis solver, click on the Tools Button (top row of WinIGS buttons), click on the grounding system symbol that contains the structural model, and then click on the **Mechanical** button.

4.5.1 Structural Dynamic Analysis Parameters

Before executing the time domain simulation, click on the toolbar button  to open the Structural Dynamic Parameters form, illustrated in Figure 4.18. Ensure that the selected options as illustrated in this Figure. Note that the **System Damping Factor** is set to 1.0. Also ensure the SDA main tool bar controls are set to the values indicated in Figure 4.19. Note that for Step 1, only the **Gravity** excitation check box is activated.

For a detailed explanation of the mechanical analysis options and parameters, please refer to chapter 1, section 1.4.

4.5.2 SDA Step 1 – Initial Condition Computation

Click on the START button in order to execute Step 1 of the dynamic analysis, which computes the initial conditions under the influence of gravity. When the simulation is completed, open the **Structural Dynamic Parameters** form (use toolbar button ). Store the system state at the end of the simulation time to be used as an initial condition for the next simulation. For this purpose click on the **Store Present State** button, and then activate the **Recall Stored State** checkbox, to force the solver to read the stored initial condition file before the next simulation. Also change the **System Damping Factor** to the value of 0.1 (a typical value for steel and aluminum buswork).

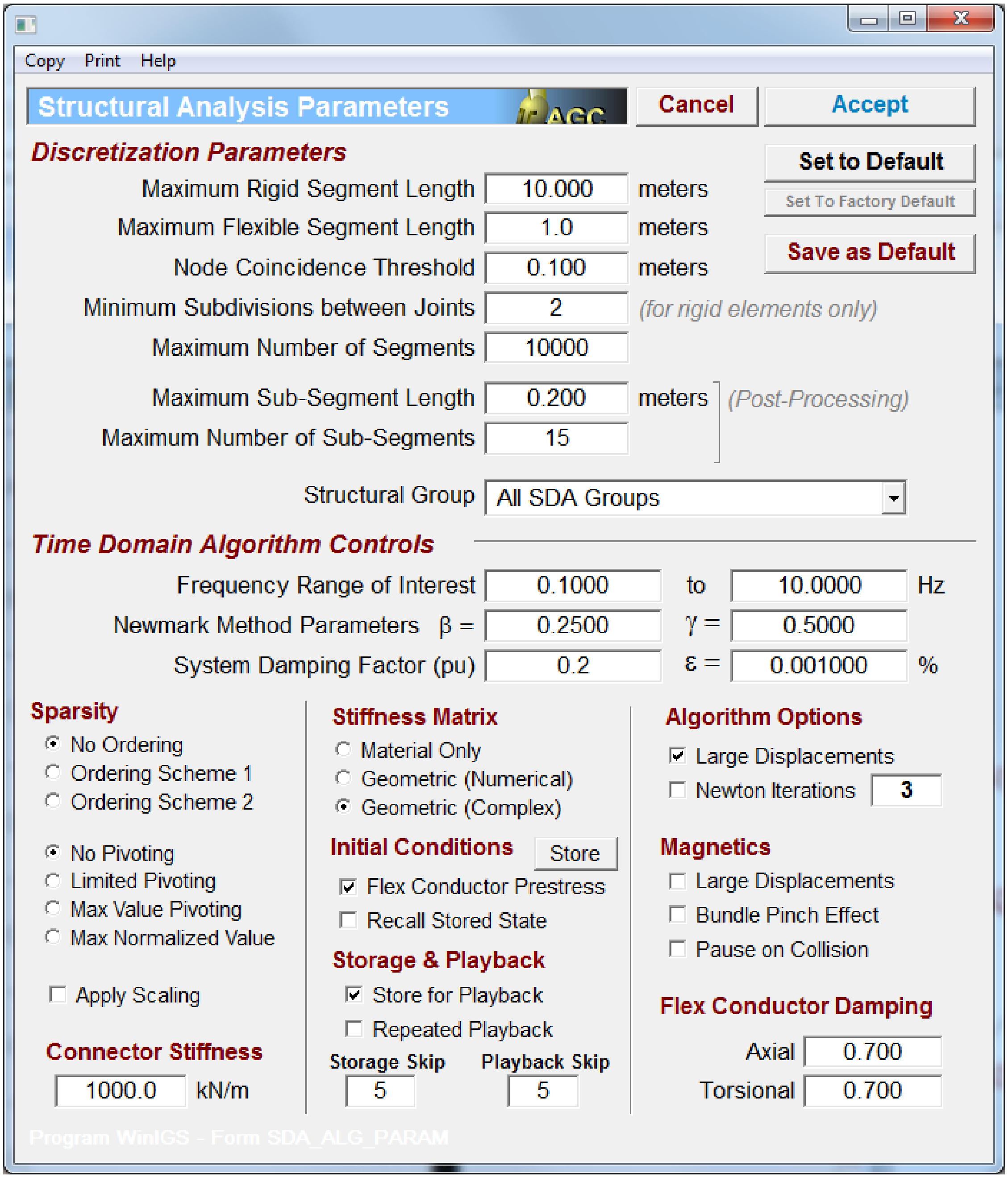


Figure 4.18: Structural Analysis Parameters

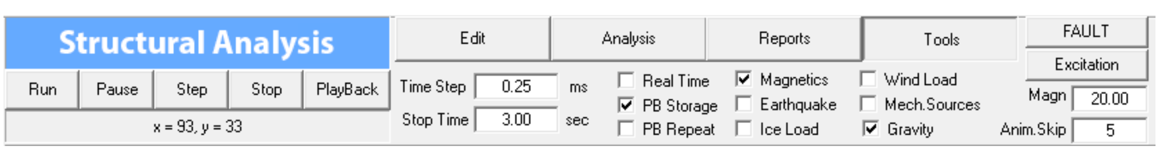


Figure 4.19: Structural Analysis Main Toolbar Controls

4.5.3 SDA Step 2 – Identification of Maximum Axial Force

In step 2 of the structural dynamic analysis, the points where maximum axial forces occur will be identified. This is achieved using the SDA "*Global Meters*". These elements continuously monitor selected quantities at all nodes of the modeled system, and keep track of where maximum values occur. At the end of the simulation the information stored by the Global Meters is used to automatically create "*Local Meters*" at the locations where maximum values occurred.

Click on the **Excitation** button of the main SDA toolbar to open the excitation parameter form, illustrated in Figure 4.20. Click on the **Magnetics** radio button to select the Fault Current parameters. Set all fault parameters as indicated in Figure 4.20.

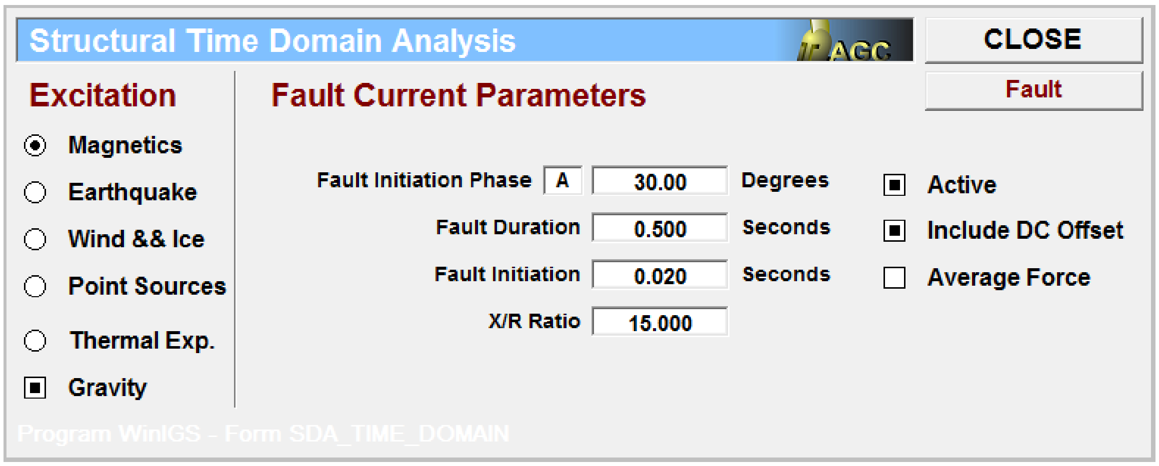
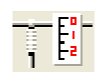


Figure 4.20: Fault Current Parameters Setup Form

Next open the meter element setup form, by clicking on the toolbar button  (or alternatively using the *Mechanical Measurement Elements* command of the *Tools* menu). This form allows the user to select the quantities of interest to be monitored by the global meters, and specific rules to be applied to the automatic meter creation procedures. The form is illustrated in Figure 4.21.

Select the quantities of interest to be:

- Displacement
- Axial Force
- Magnetic Force
- Electric Current

by clicking on the corresponding check boxes.

Select **Maxima Grouping** by **Type**. This option creates a separate meter for each group of elements of the same type. For example if the simulated system contains three types of elements: insulators aluminum pipes, and steel beams of type HSS, maximum values of the quantities of interest are separately identified for each type group. This is a recommended setting since a maximum stress occurring at an insulator may otherwise be missed if the stresses occurring on steel beams are much larger.

Set **% of Max Measured Value** to 10. This parameter sets a threshold value for each monitored quantity as a percentage of the maximum occurring value, above which meters may be created. Obviously, this parameter is meaningful only if more than one meter per quantity is to be created.

Set the **Maximum Number of Meters per Quantity** to 1.

Set the **Minimum Distance between Meters** to 10 feet.

Set the **Tracking Start Time** to 0 seconds. Setting this parameter to a positive value delays the global meter operation by the specified time. This option is useful to avoid initial artificial transients in cases where initial steady state conditions have not been previously computed and stored.

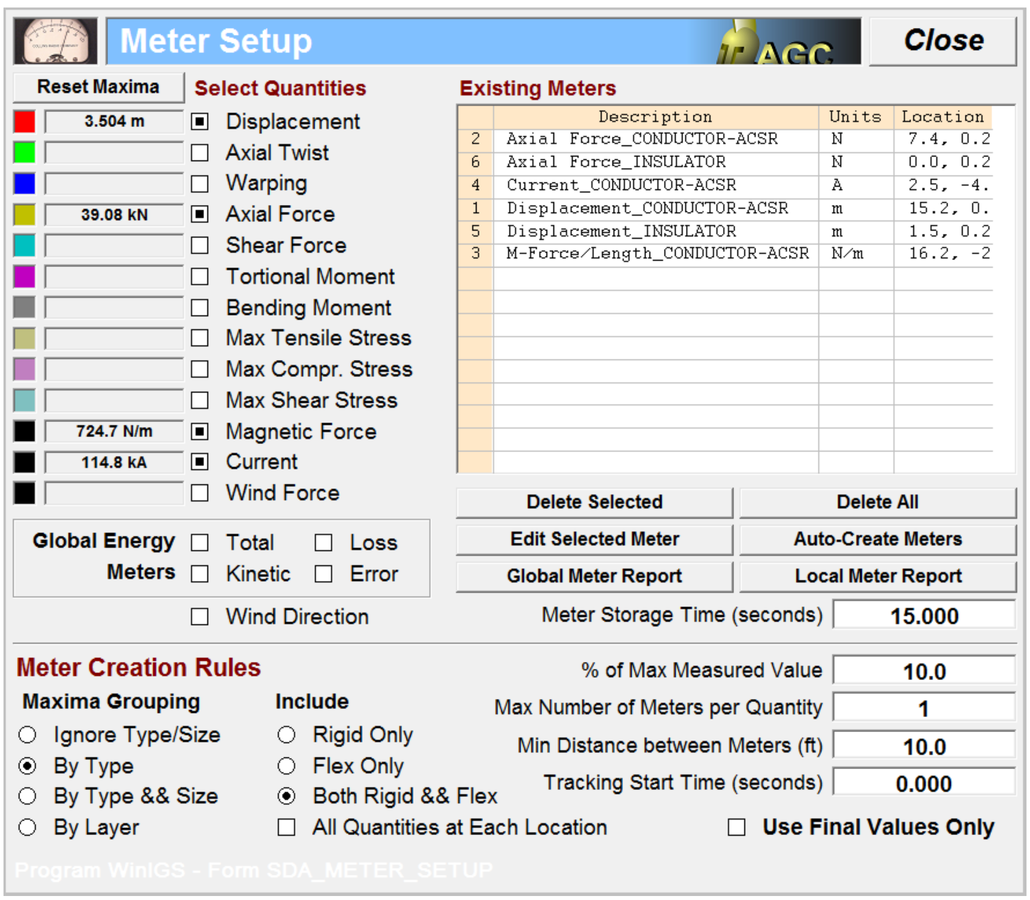


Figure 4.21: Meter Element Setup Form

Set the stop time to 3.0 seconds and click on the START button in order to execute Step 2 of the dynamic analysis.

Select the plotted waveforms to display in the plot view using the Select Plots Form, illustrated in Figure 4.22. The Select Plots form is opened by clicking on the toolbar button . This form contains two tables: a table of available waveforms (right side), and a table with waveforms that have been selected for display (left side). Note that presently, all available waveforms are generated by Global Meters. Waveforms can be added and removed from the left table in several ways. Activate the **Global**, **Stress,** and **Other** check-boxes, then click on the **Add Groups** button to generate the plot illustrated in Figure 4.23. This plot contains outputs of all global meters monitoring displacements axial forces and electric currents. Note that the **Add Groups** button puts all plots of the same quantity in the same plot frame, thus requiring only four frames to display all plot traces. Alternatively the **Add All** button places each plot trace on a separate frame.

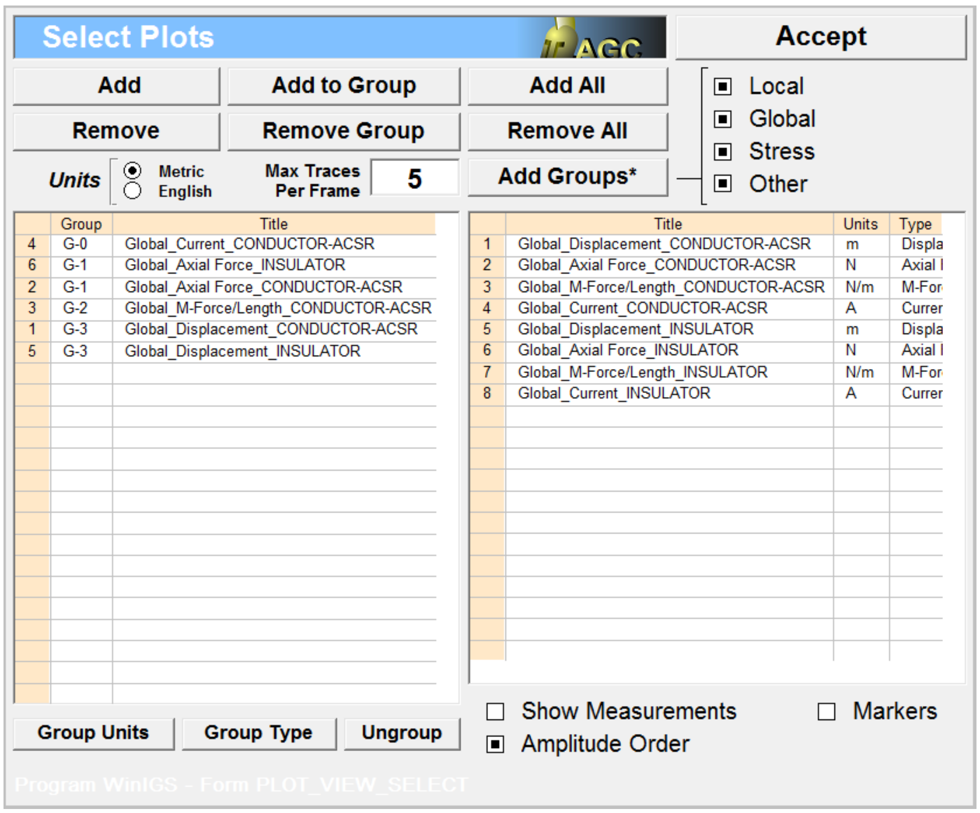


Figure 4.22: Plot Selection Form

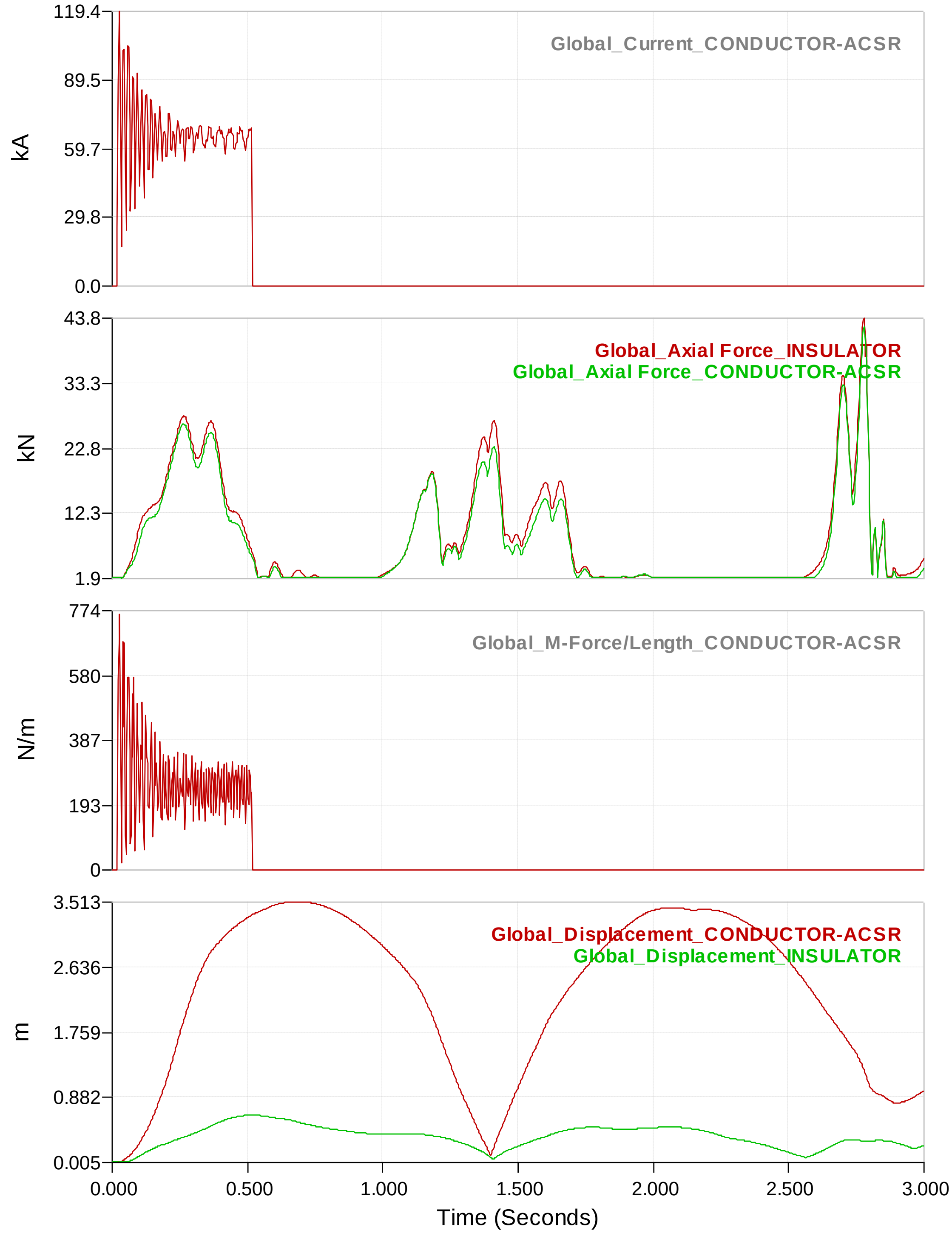
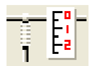


Figure 4.23: Plots Generated From Global Meters

4.5.4 SDA Step 3 – Force and Displacement at Maximum Locations

In step 3 of the structural dynamic analysis, the displacements and forces occurring at the points where maximum values occur are plotted. This is achieved by automatically creating local meters at the locations identified by Global Meters during step 2.

Once the step 2 simulation is completed, open the meter element setup form, by clicking on the toolbar button . The form is illustrated in Figure 4.21. Click on the Create Meters button of the form to automatically create local meters at the locations identified by global meters. Note that a list of the created local meters appears in the Existing Meters table of the form. Furthermore meter symbols appear at the points monitored by the local meters on all views of the simulated system. The meter locations are numbered by an index number for easy identification. Figures 4.26 and 4.27 illustrate the meter locations on a 3-D rendered view and a top view respectively. Note the meter index numbers are visible in all wireframe views (Non Rendered Views).

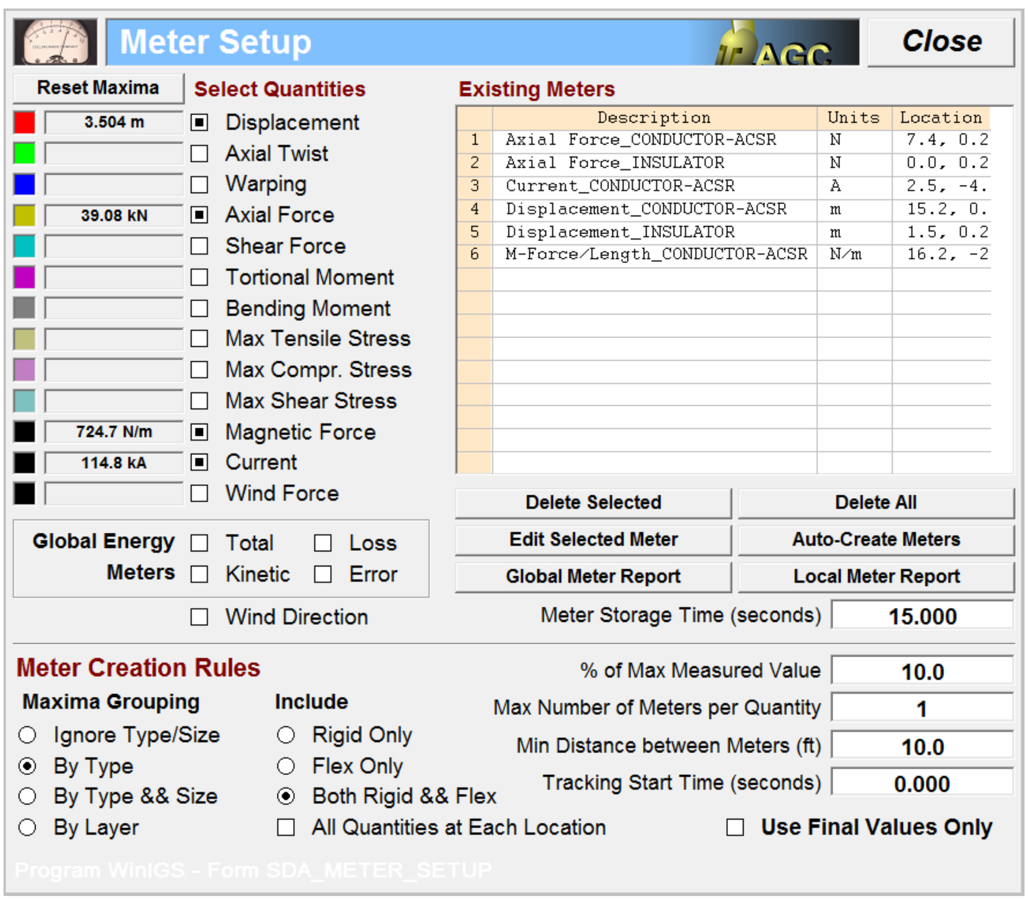


Figure 4.25: Meter Element Setup Form After Automatic Meter Creation

Set the stop time to 3.0 seconds and click on the START button in order to execute Step 3 of the dynamic analysis.

Once the simulation is completed, select the plotted waveforms to display in the plot view using the Select Plots Form. The Select Plots form is opened by clicking on the toolbar button: . De-Activate the "**Global"** check-box, then click on the **Add Groups** button to generate the plot illustrated in Figures 4.28 and 4.29. This plot contains outputs of all local meters monitoring displacements, axial forces, magnetic forces, and electric currents.

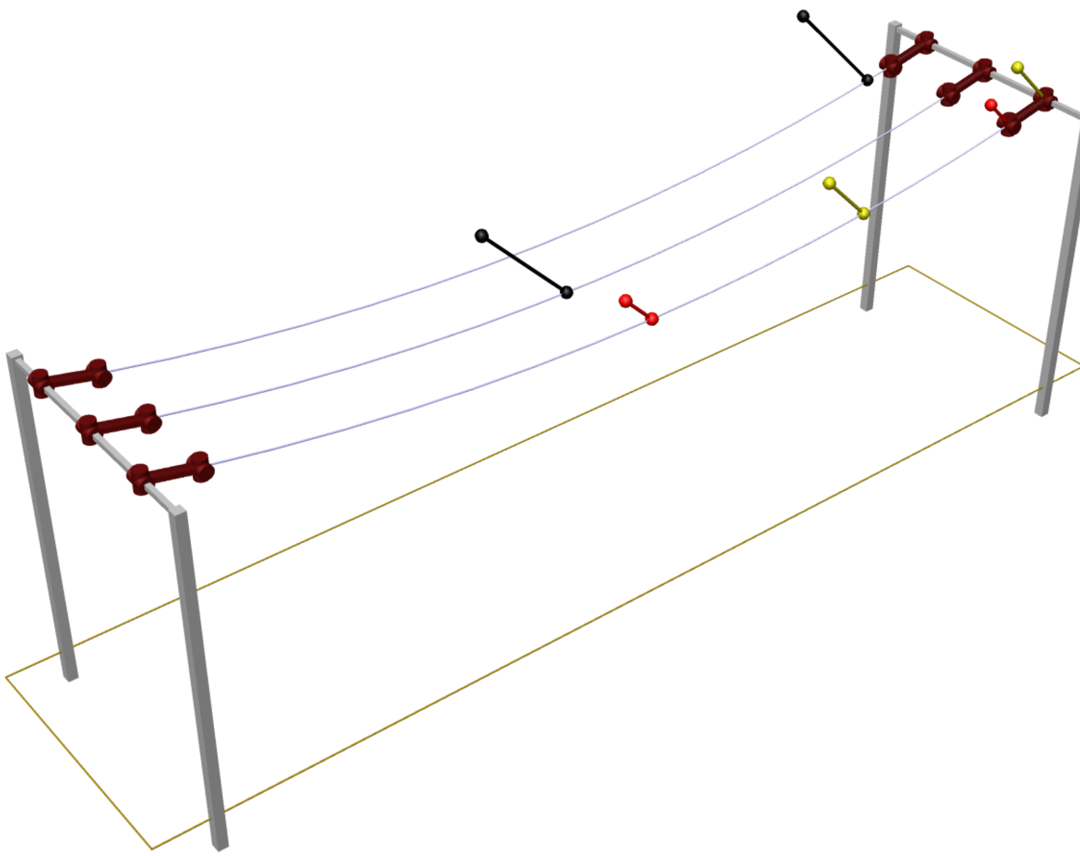


Figure 4.26: 3-D View showing Created Meter Elements

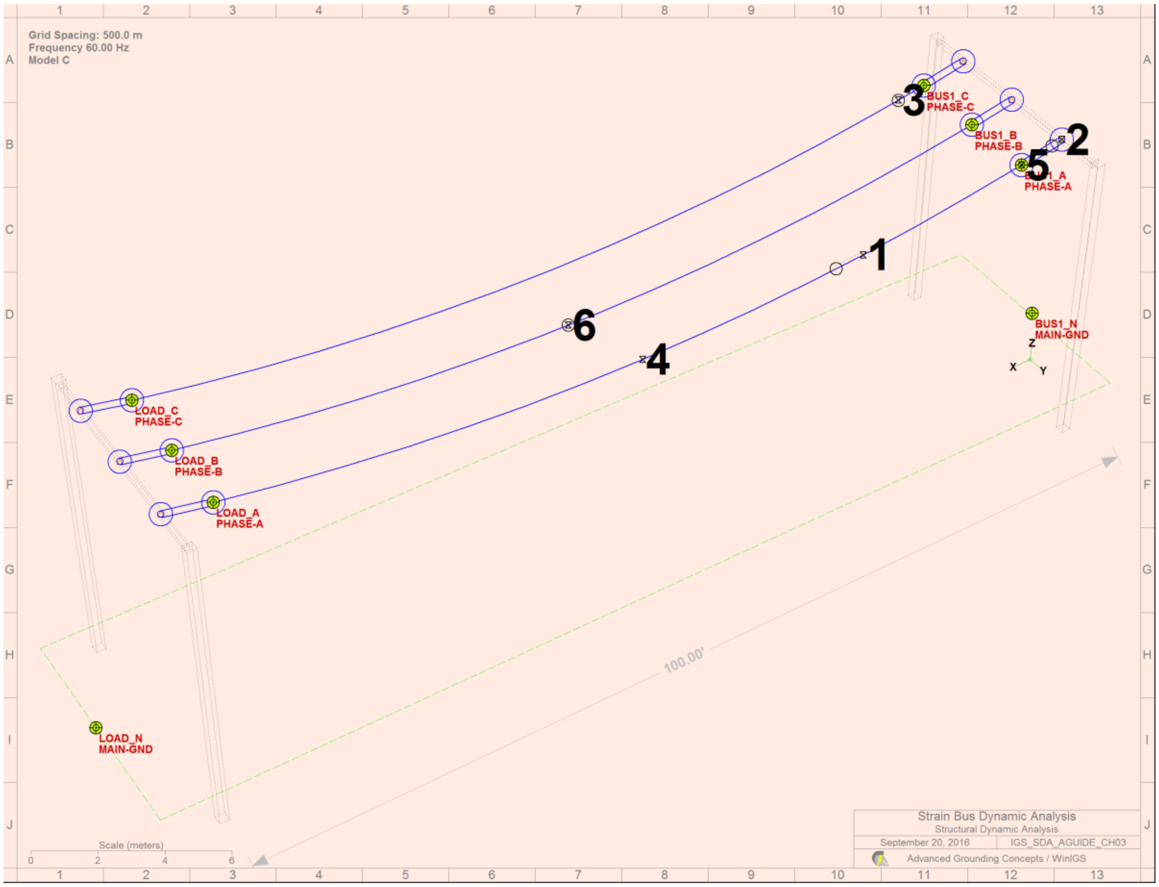


Figure 4.27: Wireframe View showing Created Meter Element Numbering

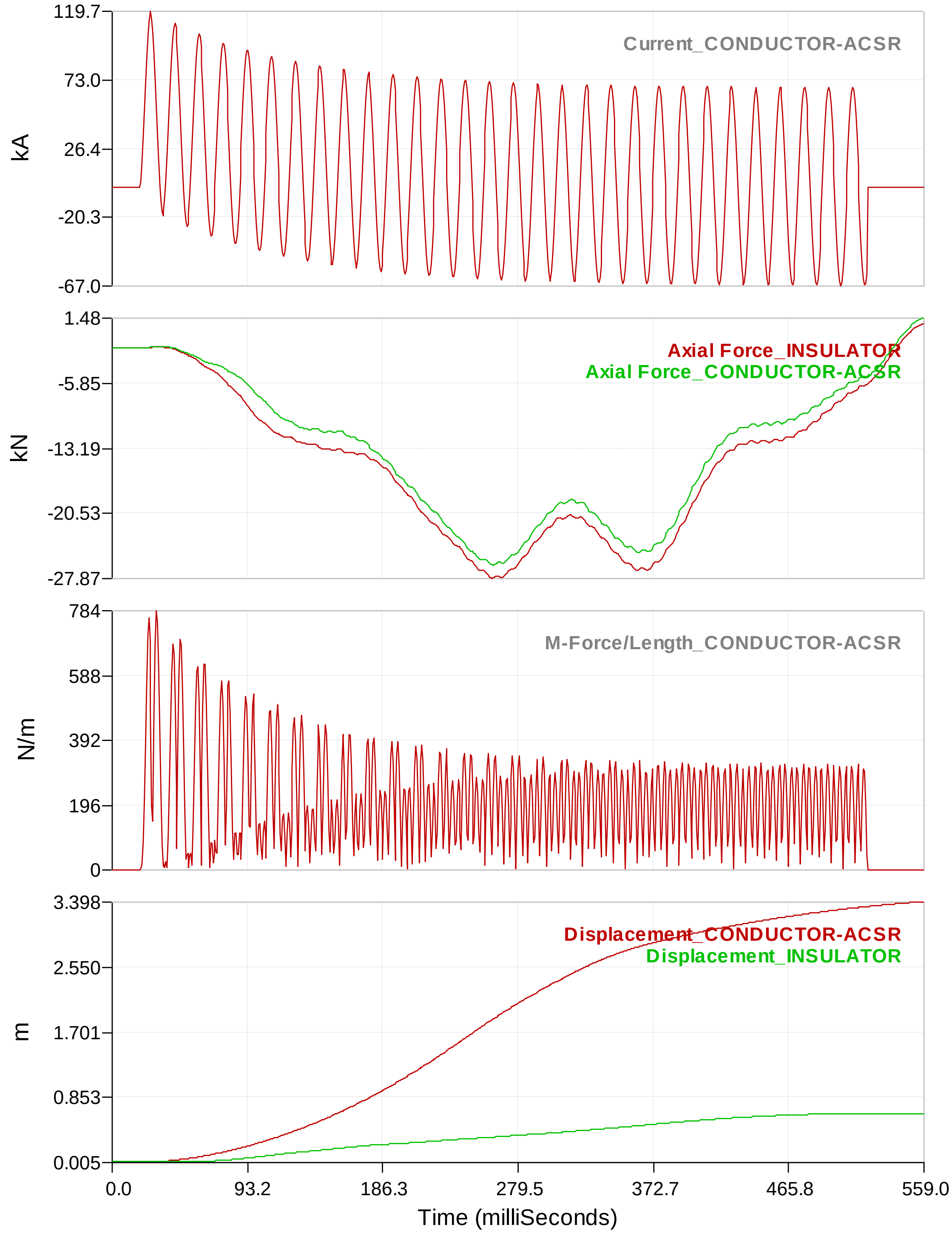


Figure 4.28: Plots Generated From Created Local Meters (First 559 milliseconds)

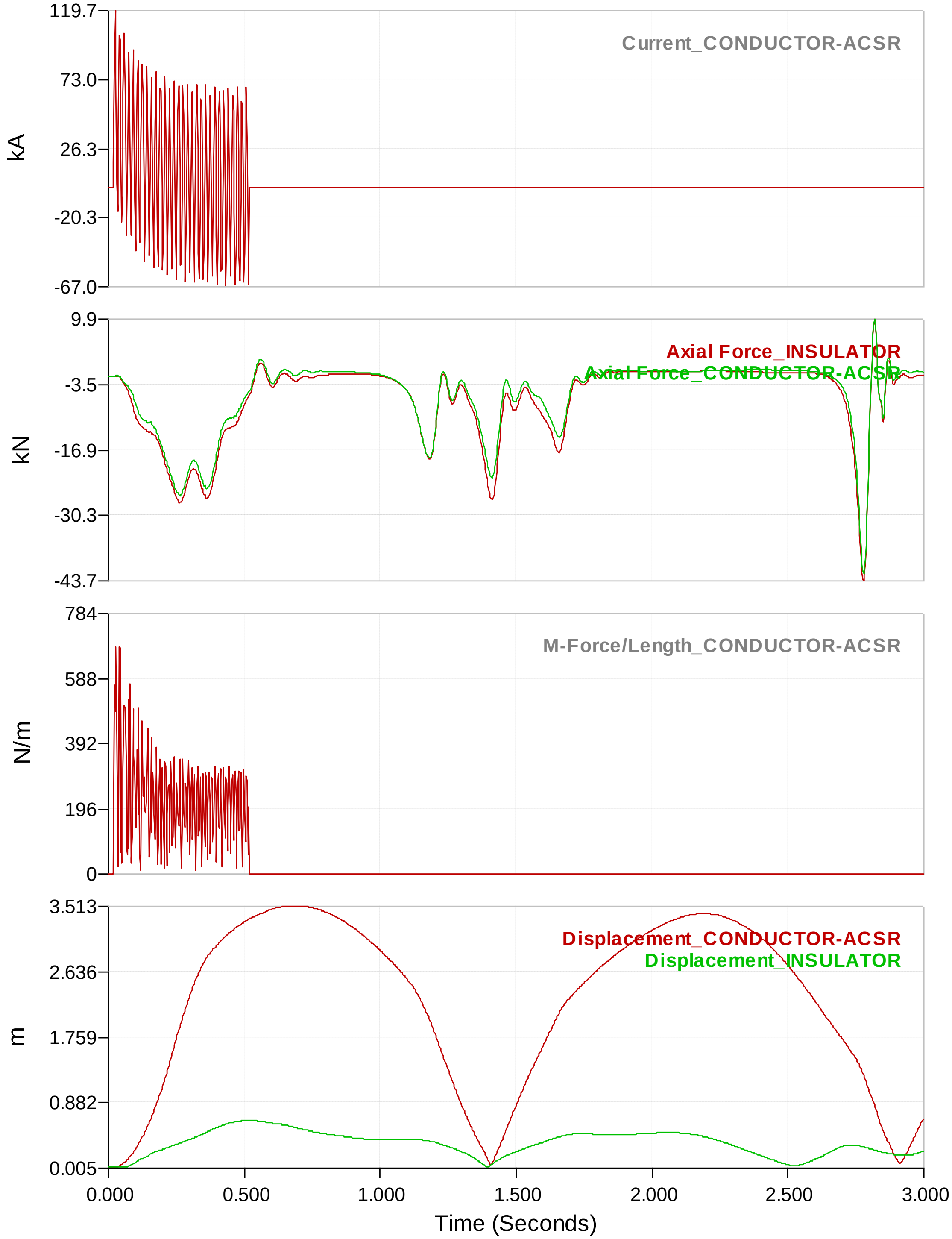


Figure 4.29: Plots Generated From Created Local Meters (First 3.0 Seconds)

4.6 Discussion

Dynamic versus Static Analysis. This example illustrates the use of the Structural Dynamic Analysis tools of WinIGS to compute the stresses occurring on a strain bus structure due to magnetic forces developing during an electrical fault. The analysis is based on a time domain simulation of the buswork dynamics. The bus conductors insulators supports, etc. are modeled using a finite element elastic beam model. This approach captures stress values occurring during oscillations, which typically exceed stresses occurring after the system reaches steady state. Thus, using dynamic analysis may identify structural failures that are missed by static analysis based techniques.

Global versus Local Meters. The stress, forces, and displacements at every point of the simulated system are monitored using global and local meters. At every time step of the simulation, each global meter scans all points of the simulated system and stores the maximum value of the quantity it monitors. It also stores the locations of *n* points where the largest *n* values occurred over the entire time duration of the simulation, where the number n is a user selected quantity. These locations are where Local Meters are automatically placed. Thus the time plot waveforms generated by global meters are not plots of a value at a single point, but an envelope curve that forms an upper bound of all the plots that can be generated by local meters at every point of the system. This approach avoids the huge storage requirement that would be necessary to store the time histories of all quantities at all points of the simulated system.

Excitation Options. In this example, the system was simulated under the influence of magnetic forces due to electrical fault currents (magnetic excitation). The effects of the structure component weights (gravity excitation) were also taken into account. Additional “excitation” options (not used in this example) include ice and wind loading, earthquakes, as well as user defined concentrated forces an moments.

5. Integrated Substation Model

5.1 Introduction

This section presents the structural dynamic analysis of a rigid three phase bus structure. This study system model includes a substation with a 3-phase rigid bus, the substation grounding system, and the major components of the 3-phase power system connected to the substation buswork. The analysis simulates the mechanical response of the buswork during an electrical fault. Specifically, the program computes the displacements and stresses occurring on the buswork components due to the magnetic forces generated during the electrical fault. Step by step instructions lead the user through opening the case data files viewing the system data, running the analysis and inspecting the results. The WinIGS data files for the example system are provided under the study case name: IGS_SDA_AGUIDE_CH05.

5.2 Inspection of System Data

Open the study case titled: IGS_SDA_AGUIDE_CH05. Use command **Open** of the **File** menu or click on the icon:  to open the existing study case data files. Note that the example study case data files are placed in the directory \IGS\DATAU during the WinIGS program installation.

Once the study case is opened, the network editor window appears showing the system single line diagram, illustrated in Figure 5.1. The system model includes seven transmission lines connecting seven equivalent 3-phase sources to the substation under study, a 500kV/230kV transformer, the substation grounding system, and a section of the 500 kV buswork.

Double click on each element of the single line diagram to examine the device parameters. Representative parameter forms for a transmission line, a source, and the 500/230 kV transformer are given in Figures 5.2, 5.3, and 5.4

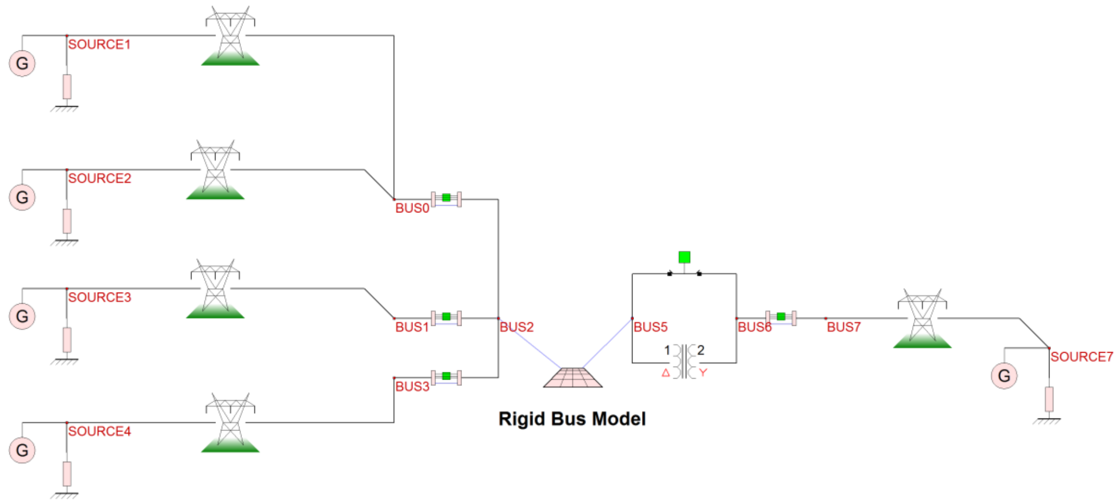


Figure 5.1 Single Line Diagram of Example System IGS_SDA_AGUIDE_CH03

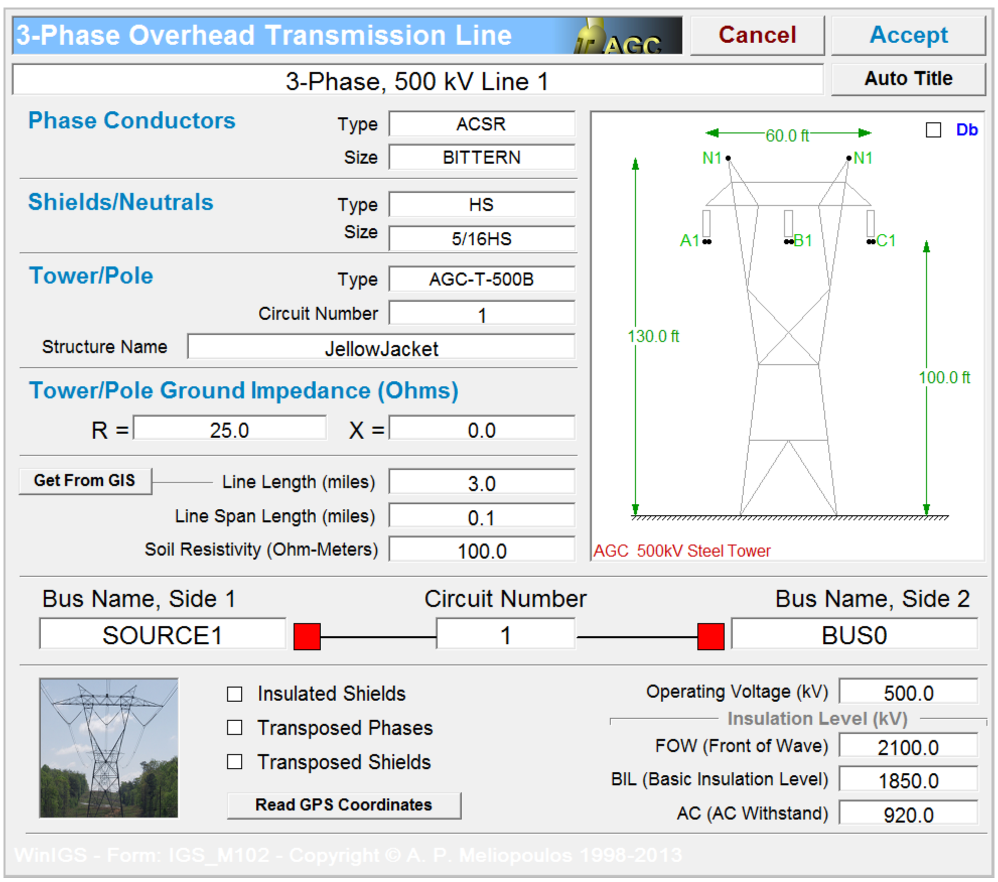


Figure 5.2: Transmission Line Parameter Form

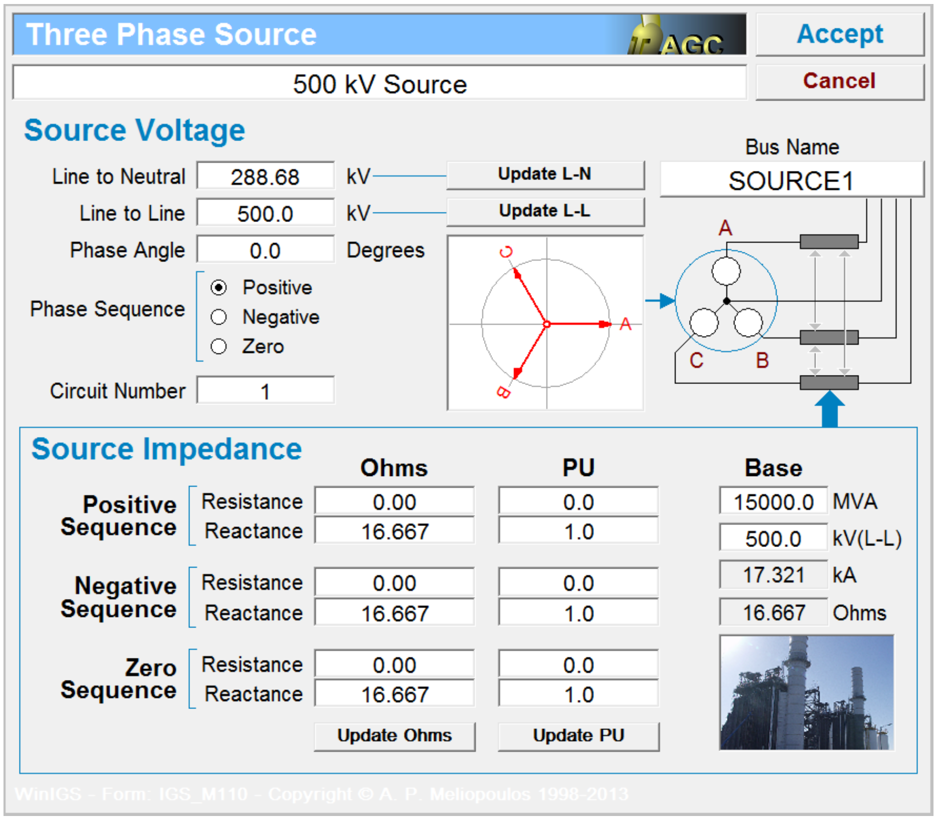


Figure 5.3: Source Parameter Form

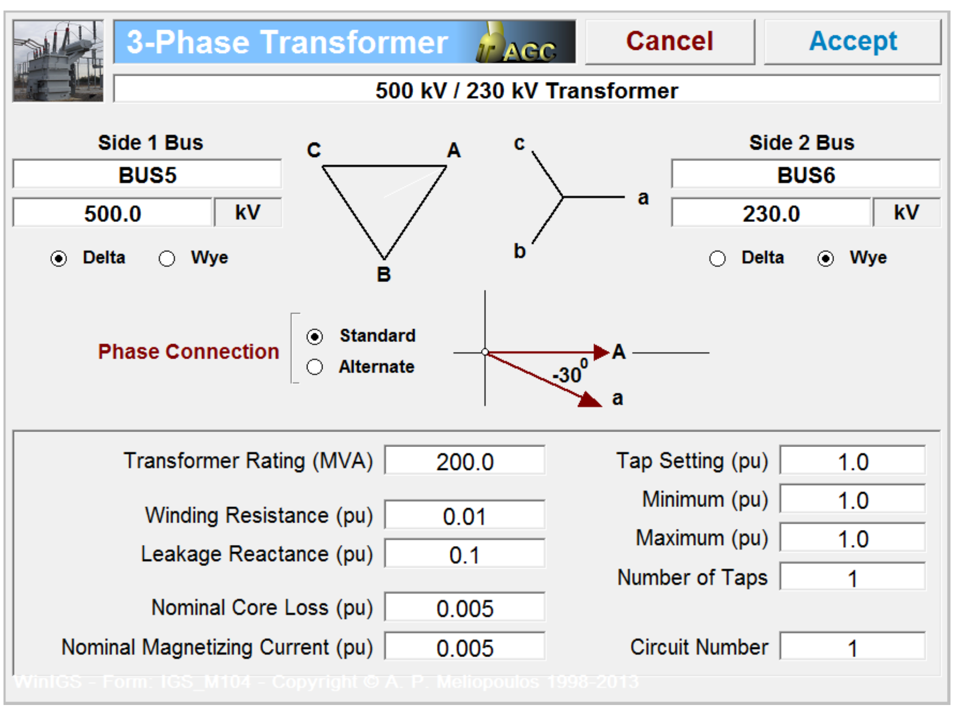
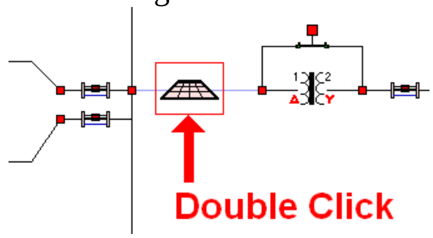
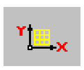


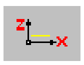
Figure 5.4: Transformer Parameter Form

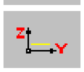
In order to inspect the 500 kV buswork model of this example, open the grounding system editor window by double clicking on the icon:

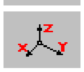


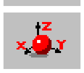
The grounding system editor provides a graphical CAD environment with extensive display and editing capabilities. The grounding system along with the modeled buswork can be displayed in top view, side view, or perspective view. Use the following toolbar buttons to switch among these viewing modes, as follows:

1  Top view (See Figure 3.5)

2  Side View

3  Side View

4  Perspective View (See Figure 3.6)

5  Rendered Perspective View (see Figure 3.7)

By default the top view of the modeled system is shown. At any view mode you can zoom using the mouse wheel and pan by moving the mouse while holding down the mouse right button. In the perspective view mode, you can also rotate the view point by holding down both the keyboard Shift key and the right mouse button.

The modeled system consists includes a 3-phase rigid bus (shown in blue in the top view) two 3-Phase breakers (green) and two transmission line end support structures (gold). Note that the breakers and Line supports are represented by “Non-SDA” elements. Non-SDA elements are ignored by the structural analysis solver.

Figure 3.8 illustrates the bus support structure detail. The rigid conductors are supported by insulators, which are supported by vertical steel beams. A support element (triangle symbol) at the base of each vertical steel beam represents the attachment of the steel beam bottom to the concrete foundation. It is an essential requirement for any system to be simulated that all SDA elements (support beams conductors, insulators etc) to be properly supported by appropriately placed support elements. Specifically, support elements should ensure no SDA element can freely move or rotate. Click on each element comprising the bus support system to inspect the element parameters. Figures 3.9 through 3.12 illustrate the parameters of the major elements.

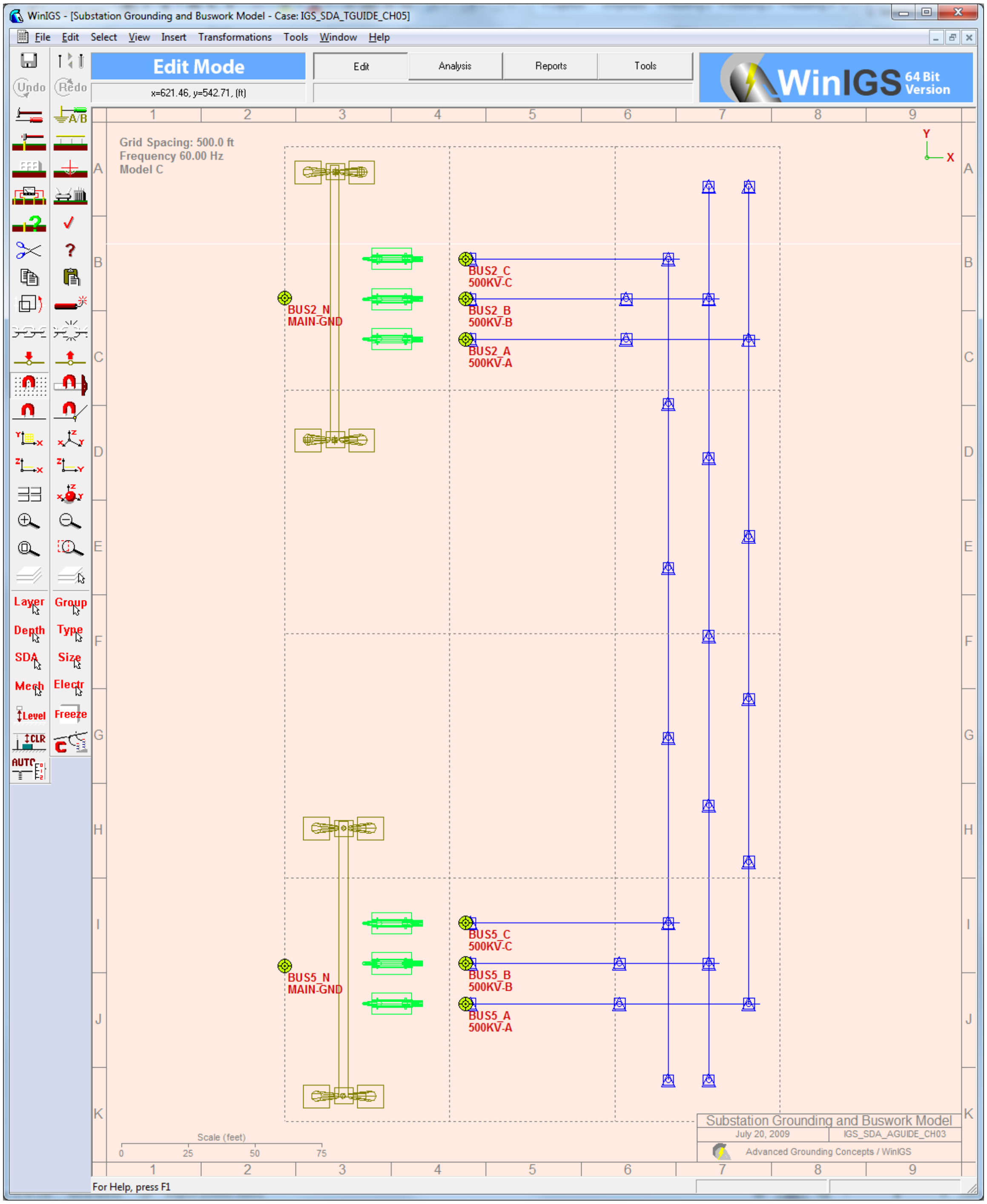


Figure 5.5: 3-Phase Rigid Bus Model – Top View

In order to compute the magnetic forces that develop on the bus conductors during electrical faults, the electric current through the bus conductors must be evaluated first. For this purpose the model includes a physical representation of the conductive paths along each phase of the bus and the connections to the external network. Note that the connections of the bus conductor ends to the external network components (transmission lines, transformer etc) are made via eight interface nodes, represented by the symbol , end labeled BUS2_A, BUS2_B, BUS2_C, BUS2_N and BUS5_A, BUS5_B, BUS5_C, BUS5_N. Each of these interface nodes produces a connecting terminal on the network view (single line diagram) which are attached to the appropriate devices. The connections at each terminal node of the single line diagram can be inspected by double clicking on the node. Figure 3.13 illustrates the inspection window for the connections between the modeled buswork and the substation transformer. This window is opened by double clicking on the BUS5 node.

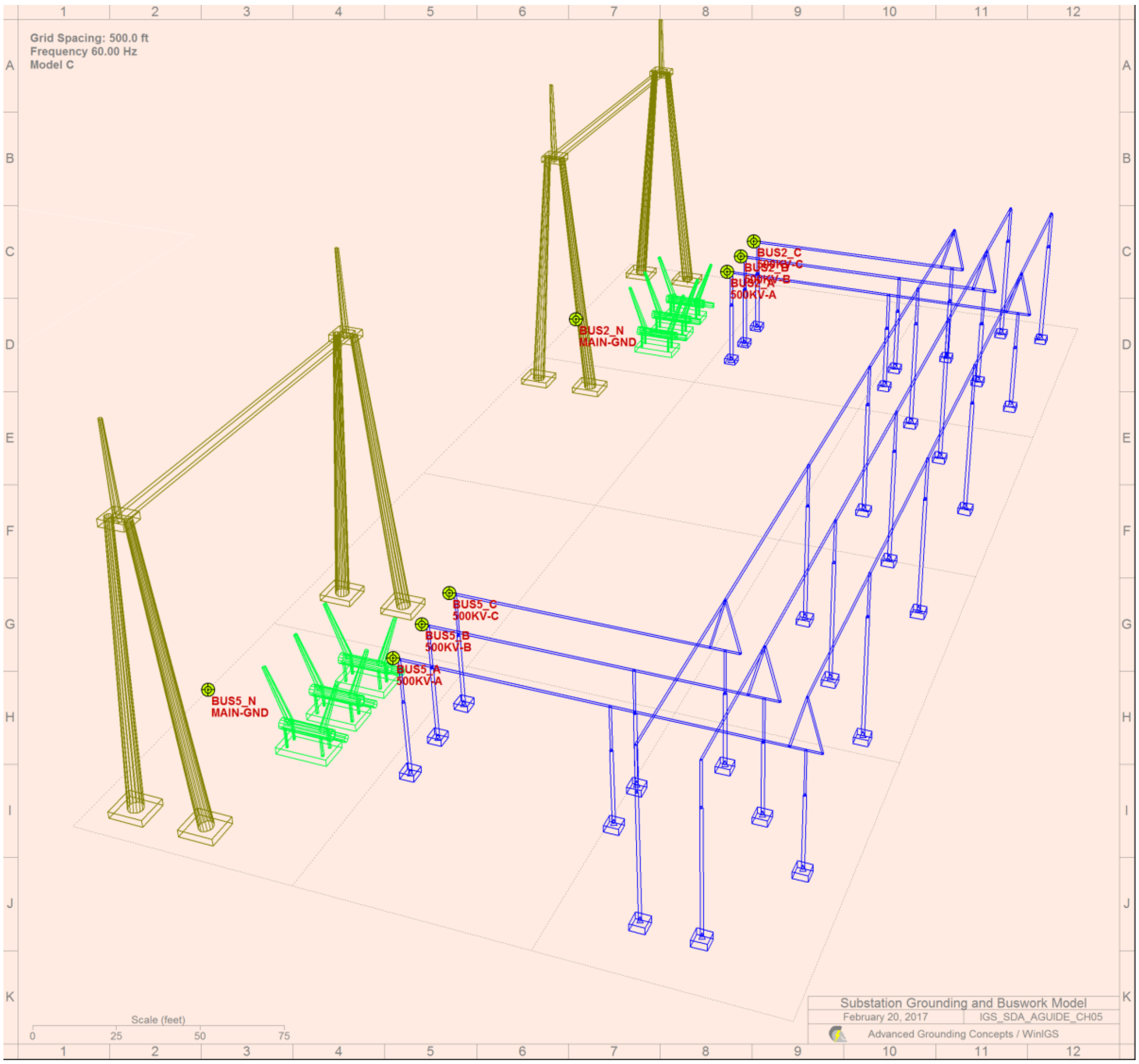


Figure 5.6: 3-Phase Rigid Bus Model – Perspective View

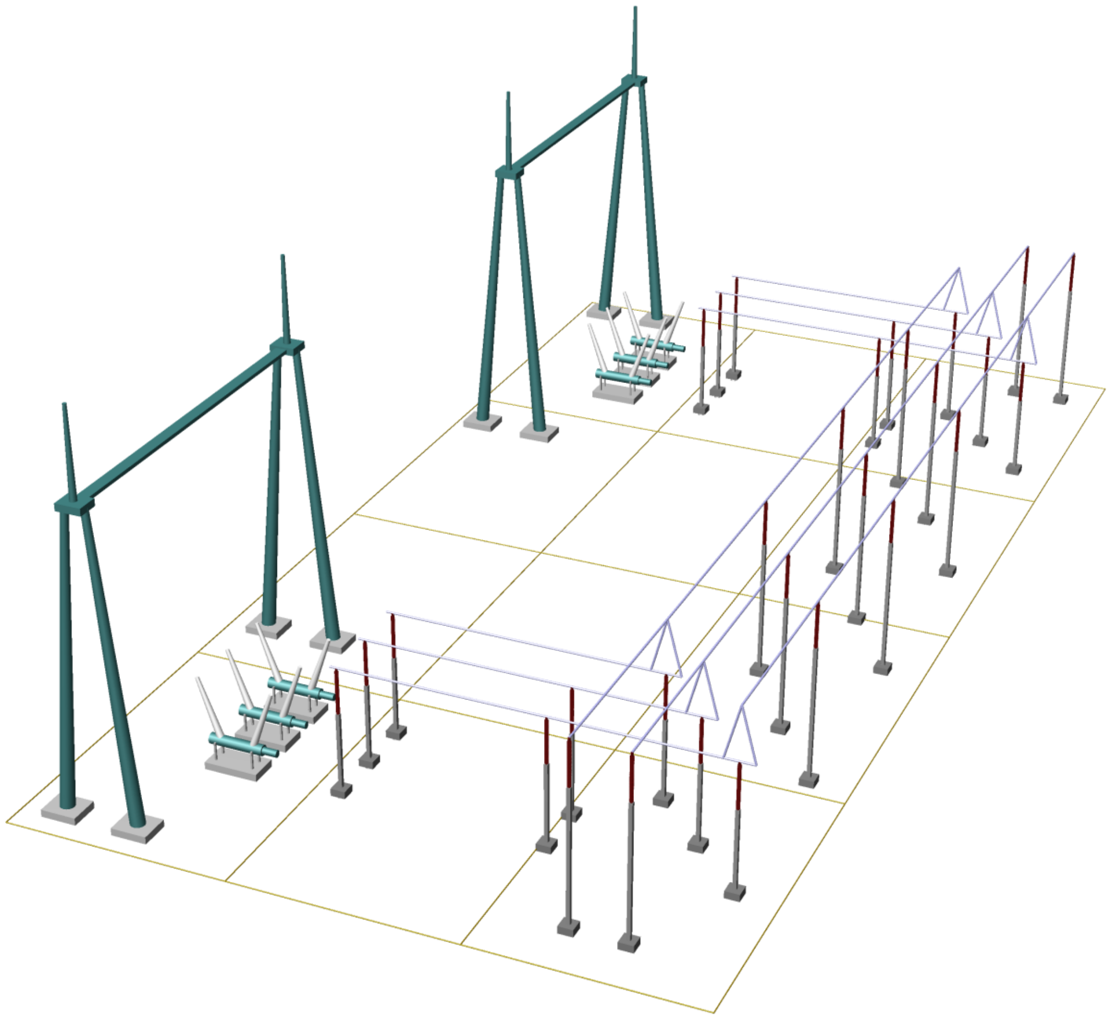


Figure 5.7: 3-Phase Rigid Bus Model – Rendered Perspective View

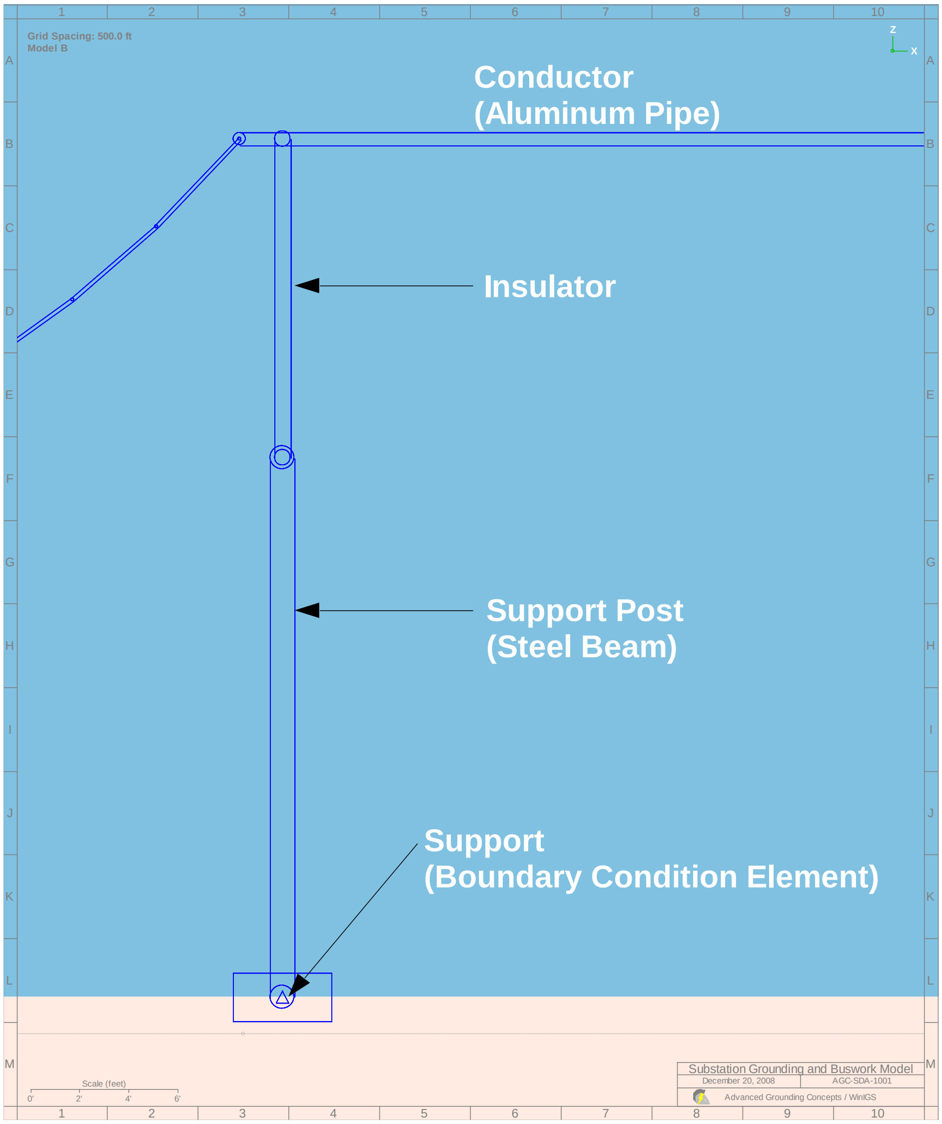


Figure 5.8: Bus Model – Side View Detail

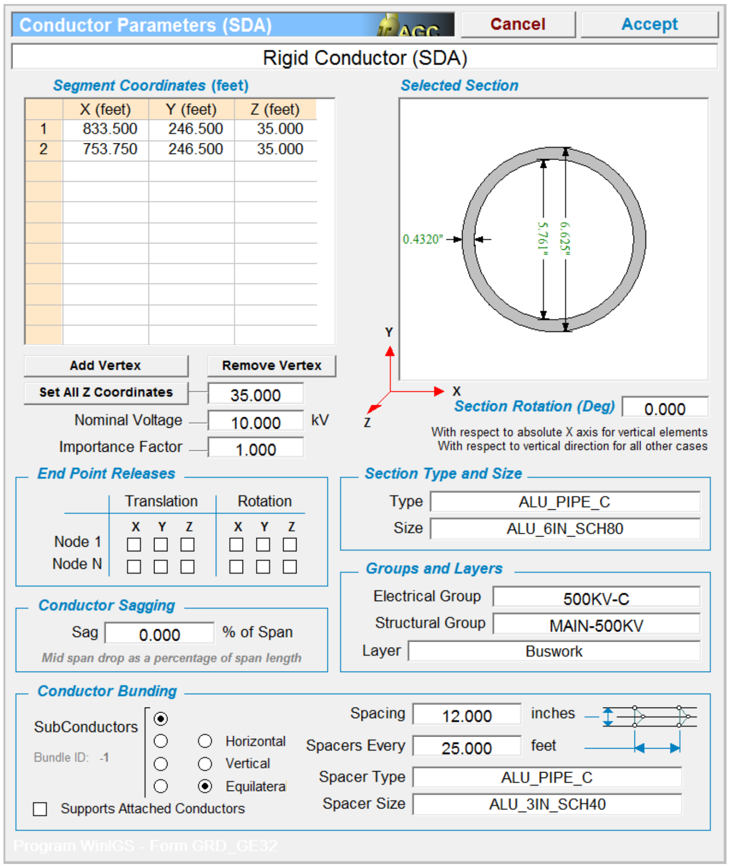


Figure 4

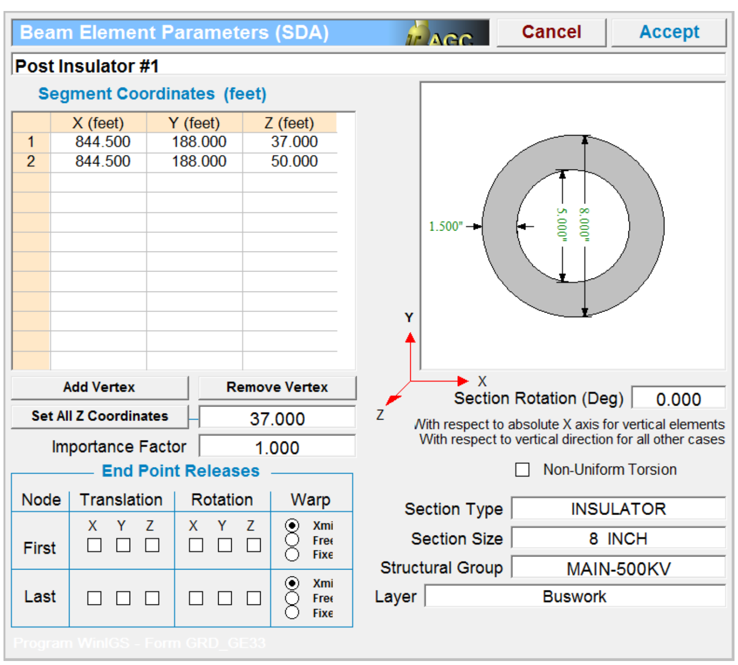


Figure 5.10: Insulator Model Parameter Form

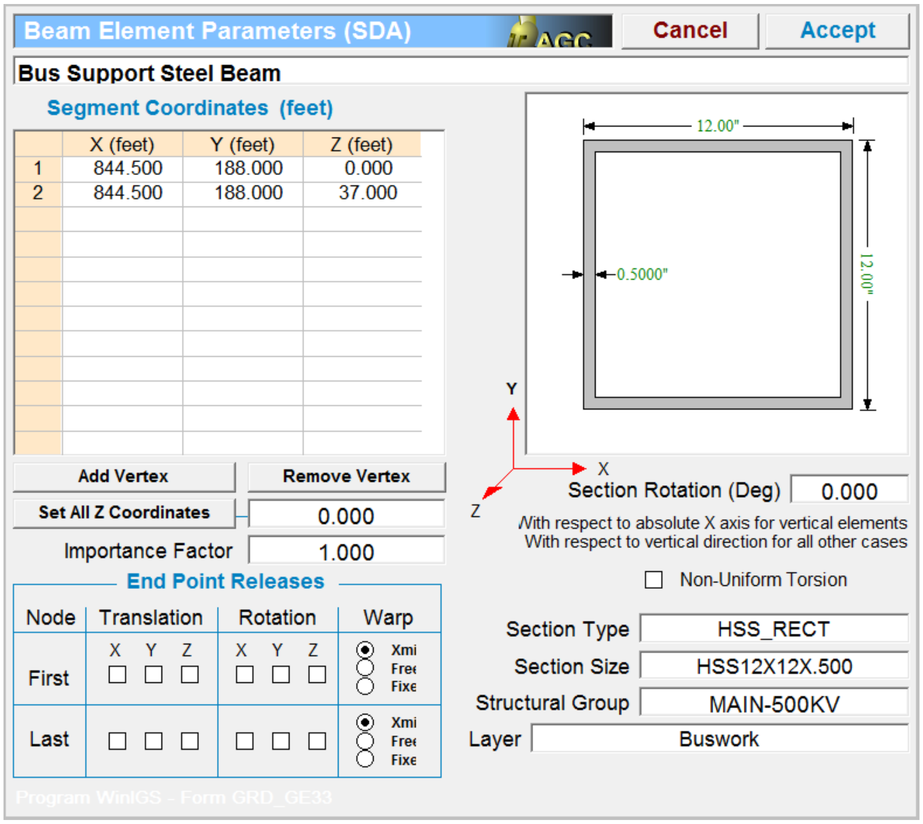


Figure 5.11: Steel Beam Support Parameter Form

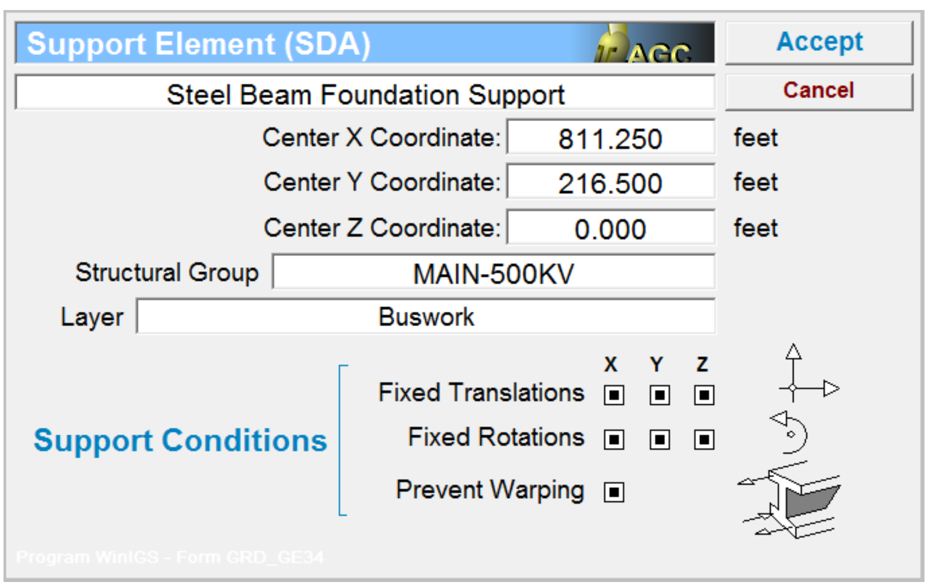


Figure 5.12: Support Element Parameter Form

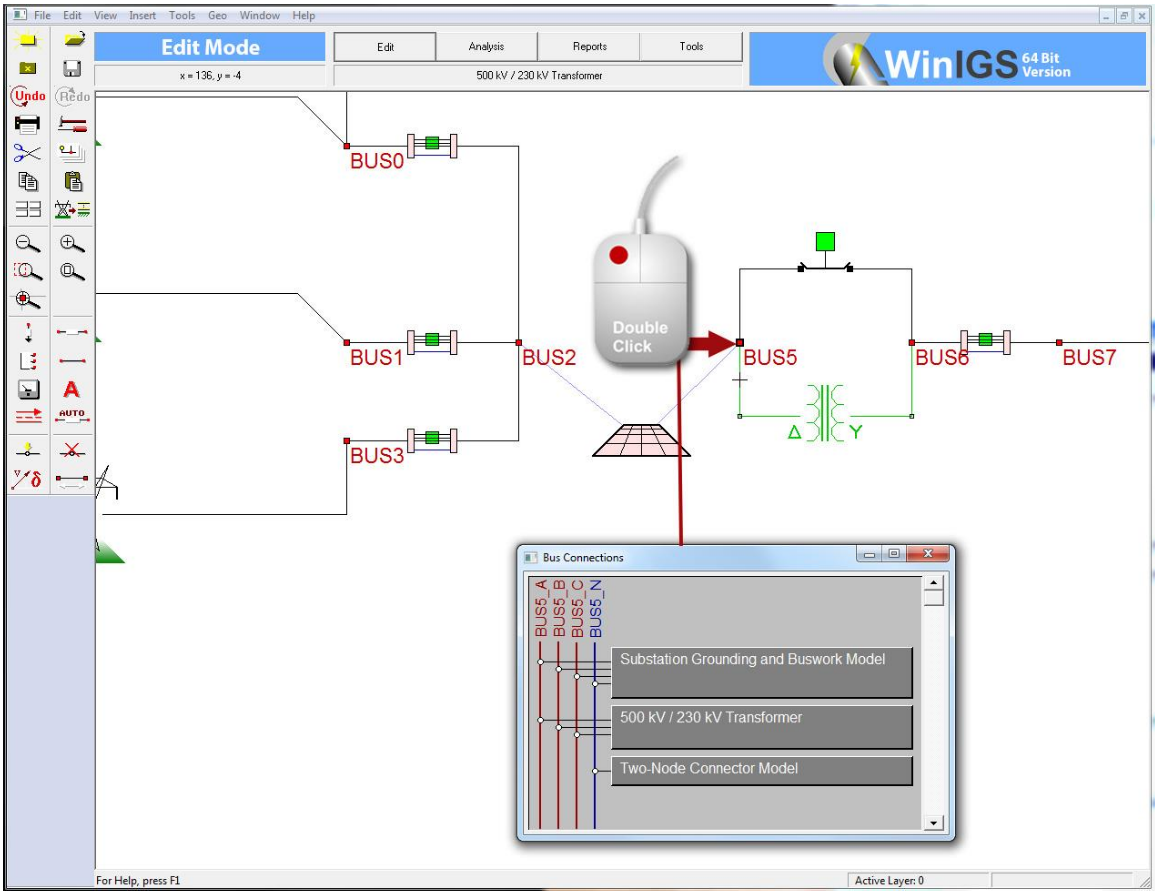


Figure 5.13: Inspection of Bus Connections

5.3 Electrical Fault Analysis

The fault analysis of this system comprises of two steps.

- Step 1 – Equivalent Impedance Analysis
- Step 2 – Electrical Fault Analysis

The fault analysis step can be performed using any if the fault analysis options provided by WinIGS. In this example a three phase fault on the 500 kV bus (BUS5) is selected. The electrical fault analysis results are automatically picked up by the structural analysis solver in order to compute the resulting magnetic forces on the modeled buswork.

In addition to the fault current levels, the X/R ratio at the fault location is required in order to estimate the DC offset of the current waveforms. The X/R ration is obtained by performing Port Thevenin Equivalent analysis at the fault location, as follows:

- Click on the **Analysis** button
- From the pull-down list control select the "**Port Thevenin Equivalent**" option
- Click on the **Run** button to open the equivalent impedance analysis form, illustrated in Figure 3.15.
- Select the **3–Phase Bus Port** radio button, and the **Positive Sequence** option.
- Click on the **Execute** button, and note the reported X/R ratio.

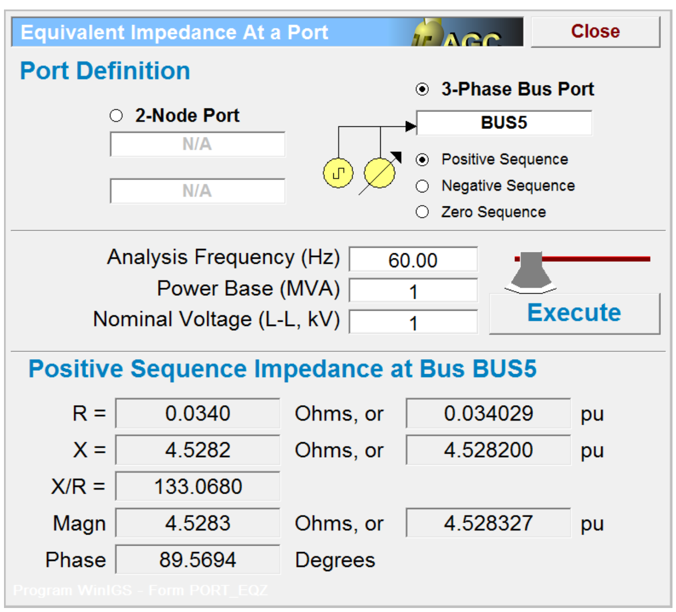


Figure 5.14: Equivalent Impedance Report

The procedure for performing fault analysis is as follows:

- Click on the **Analysis** button

- From the pull-down list control select the "**Fault Analysis**" option
- Click on the **Run** button to open the fault analysis option form.

Note that all these controls are located along the top side of the main program window frame.

The fault analysis option form is illustrated in Figure 3.15. Ensure that the selected option are as illustrated in this Figure. Specifically, select **Fault at a Bus** radio button, Set **Fault Type** to **Three Phase Fault,** select **Faulted Phases / Lines** to be **Phase A, Phase B,** and **Phase C**, and then click on the **Execute** button.

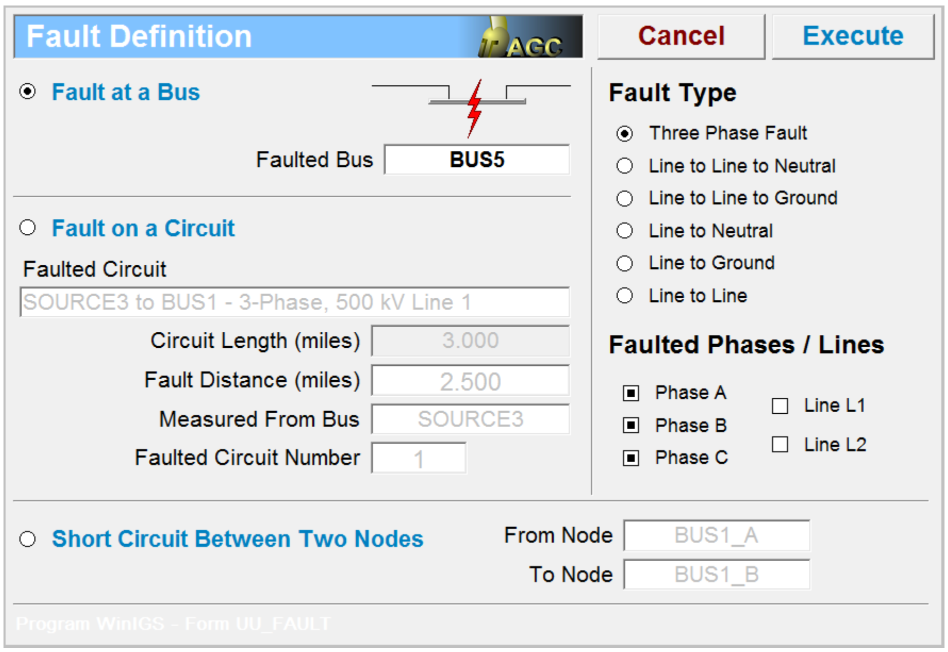


Figure 5.15: Fault Analysis Options

Once the analysis is completed, a pop-up window appears indicating the completion of the analysis. This window, illustrated in Figure 3.16, also displays the computed fault current. Click on the **Close** button to close this window, and then click on the **Reports** button to enter into the report viewing mode.

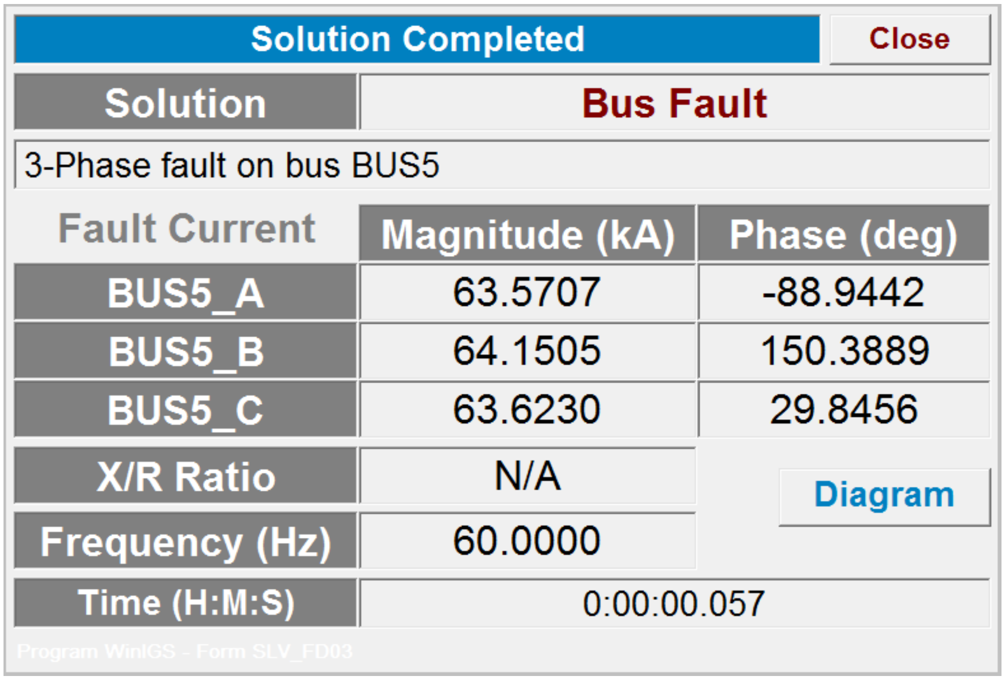


Figure 5.16: Fault Analysis Completion Report

5.4 Inspection of Fault Analysis Results

While in Reports mode, a set of "*radio buttons*" appears along the top of the main program window frame, which allows selection of the report type. From these buttons, select the **Graphical I/O report**, and then double click on the grounding system icon to view the Device Voltage and Current Report. This report is illustrated in Figure 3.17. Note that the current circulating through the modeled bus is about 62 kA.

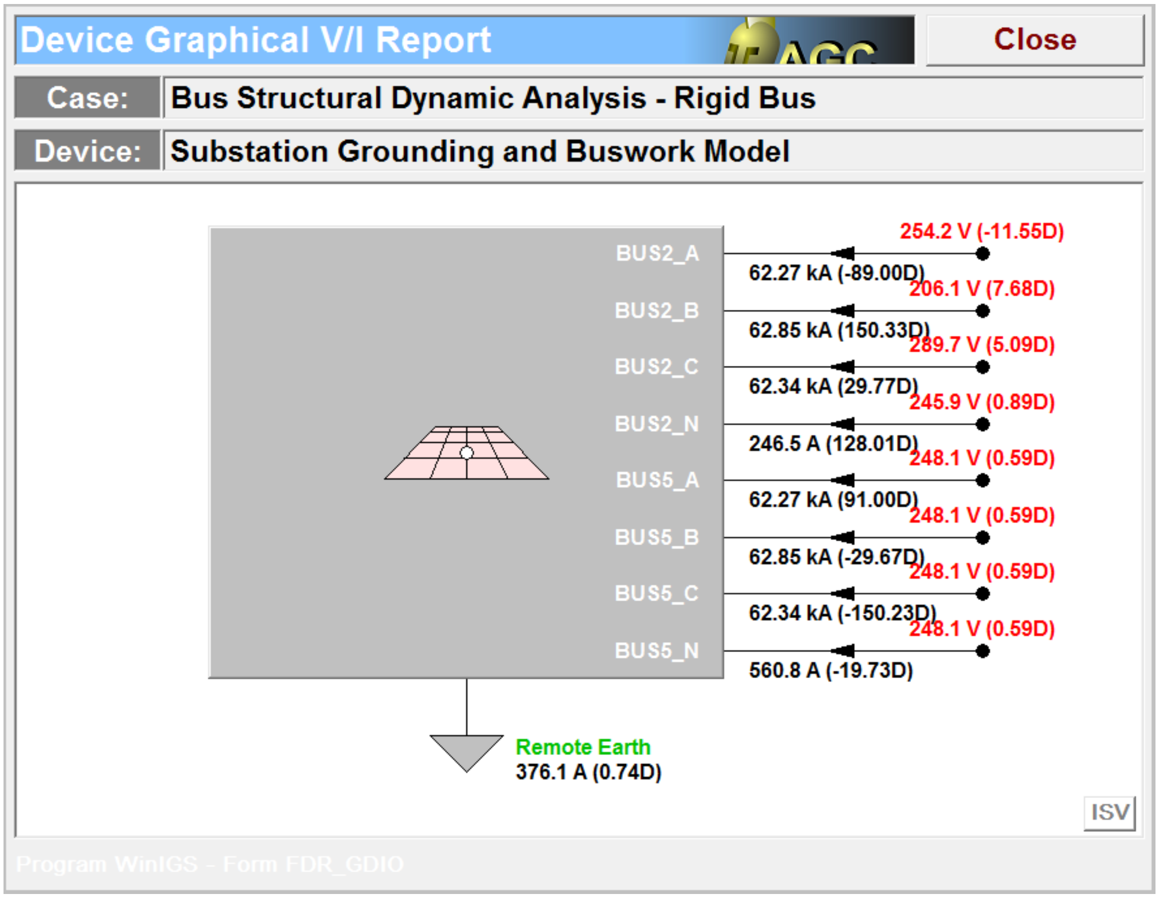


Figure 3.17: Device Voltage and Current Report

5.5 Structural Dynamic Analysis (SDA)

The structural dynamic analysis is performed in three steps:

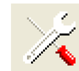
Step 1: Perform analysis with gravity excitation only. The objective of this step is to compute and store the appropriate steady state conditions. In order to speed up the decay of the oscillations, it is recommended that the system damping ratio is set to a high value (e.g. 1.0).

Step 2: Perform analysis with both gravity and magnetic force excitation. The objective of this step is to identify the locations where maximum stresses occur during the simulated electrical fault. The initial conditions are obtained from the results of Step 1. The system damping ratio is set to a typical value for rigid bus structures (e.g. 0.1).

Step 3: Repeat analysis with both gravity and magnetic force excitation. The objective of this step is to generate plots of stress versus time at the points of maximum stress identified in Step 2. The initial conditions are again obtained from the results of Step 1.

In order to access the Structural Dynamic Analysis solver, click on the Tools Button (top row of WinIGS buttons), click on the grounding system symbol that contains the structural model, and then click on the **Mechanical** button.

3.5.1 Structural Dynamic Analysis Parameters

Before executing the time domain simulation, click on the toolbar button  to open the Structural Dynamic Parameters form, illustrated in Figure 3.18. Ensure that the selected options as illustrated in this Figure. Note that the **System Damping Factor** is set to 1.0. Also ensure the SDA main tool bar controls are set to the values indicated in Figure 3.19. Note that for Step 1, only the **Gravity** excitation check box is activated.

For a detailed explanation of the mechanical analysis options and parameters, please refer to chapter 1, section 1.4.

3.5.2 SDA Step 1 – Initial Condition Computation

Click on the START button in order to execute Step 1 of the dynamic analysis, which computes the initial conditions under the influence of gravity. When the simulation is completed, open the **Structural Dynamic Parameters** form (use toolbar button ). Store the system state at the end of the simulation time to be used as an initial condition for the next simulation. For this purpose click on the **Store Present State** button, and then activate the **Recall Stored State** checkbox, to force the solver to read the stored initial condition file before the next simulation. Also change the **System Damping Factor** to the value of 0.1 (a typical value for steel and aluminum buswork).

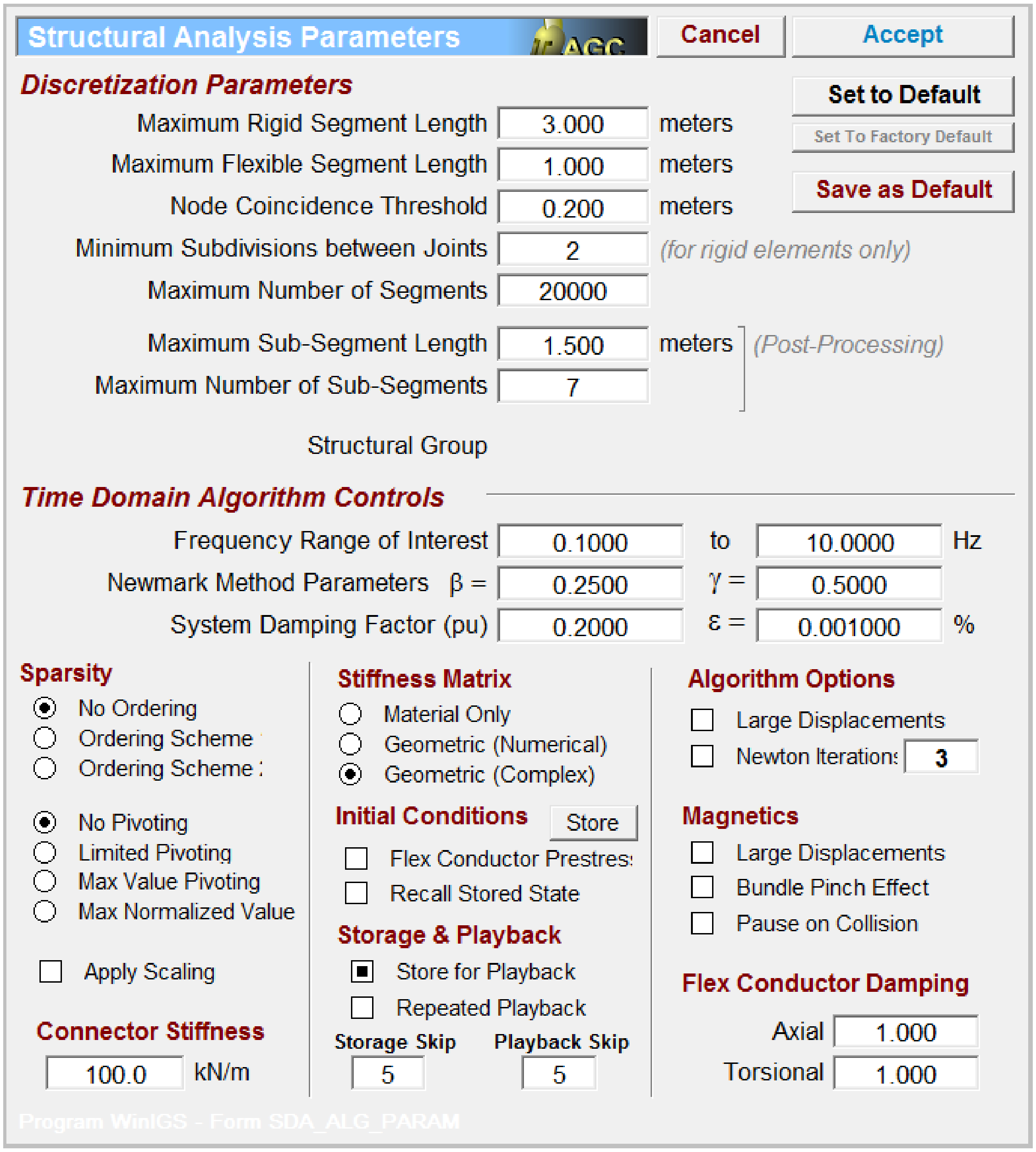


Figure 5.18: Structural Analysis Parameters

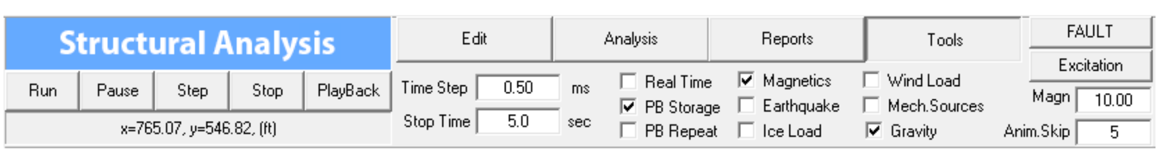


Figure 5.19: Structural Analysis Main Toolbar Controls

5.5.3 SDA Step 2 – Identification of Maximum Stress Points

In step 2 of the structural dynamic analysis, the points where maximum stresses occur will be identified. This is achieved using the SDA "*Global Meters*". These elements continuously monitor selected quantities at all nodes of the modeled system, and keep track of where maximum values occur. At the end of the simulation the information stored by the Global Meters is used to automatically create "*Local Meters*" at the locations where maximum values occurred.

Click on the **Excitation** button of the main SDA toolbar to open the excitation parameter form, illustrated in Figure 3.20. Click on the **Magnetics** radio button to select the Fault Current parameters. Set all fault parameters as indicated in Figure 3.20.

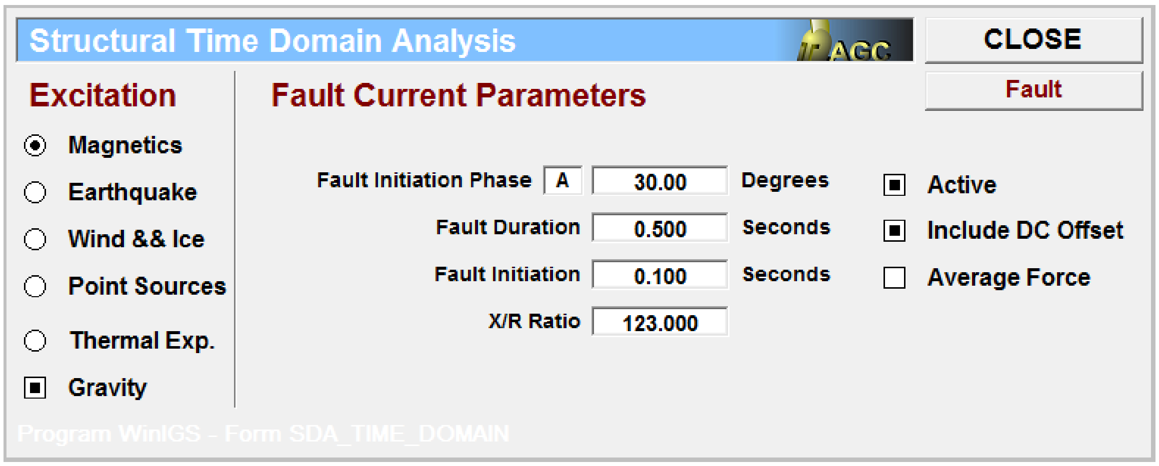
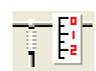


Figure 5.20: Fault Current Parameters Setup Form

Next open the meter element setup form, by clicking on the toolbar button  (or alternatively using the *Mechanical Measurement Elements* command of the *Tools* menu). This form allows the user to select the quantities of interest to be monitored by the global meters, and specific rules to be applied to the automatic meter creation procedures. The form is illustrated in Figure 3.21.

Select the quantities of interest to be:

- Displacement
- Maximum Tensile Stress
- Maximum Compression Stress
- Maximum Shear Stress
- Magnetic Force
- Electric Current

by clicking on the corresponding check boxes.

Check the box **All Quantities at Each Location** to create meters of all selected quantities at each location where maximum value of any selected quantity occurs.

Select **Maxima Grouping** by **Type**. This option creates a separate meter for each group of elements of the same type. For example if the simulated system contains three types of elements: insulators aluminum pipes, and steel beams of type HSS, maximum values of the quantities of interest are separately identified for each type group. This is a recommended setting since a maximum stress occurring at an insulator may otherwise be missed if the stresses occurring on steel beams are much larger.

Set **% of Max Measured Value** to 10. This parameter sets a threshold value for each monitored quantity as a percentage of the maximum occurring value, above which meters may be created. Obviously, this parameter is meaningful only if more than one meter per quantity is to be created.

Set the **Maximum Number of Meters per Quantity** to 1.

Set the **Minimum Distance between Meters** to 10 feet.

Set the **Tracking Start Time** to 0 seconds. Setting this parameter to a positive value delays the global meter operation by the specified time. This option is useful to avoid initial artificial transients in cases where initial steady state conditions have not been previously computed and stored.

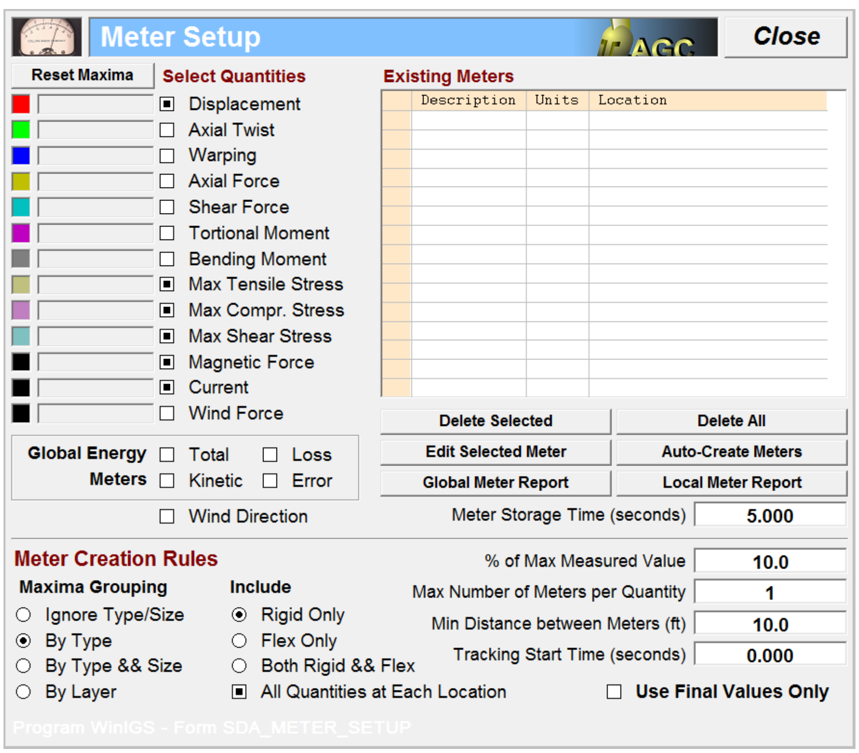


Figure 5.21: Meter Element Setup Form

Set the stop time to 1.25 seconds and click on the START button in order to execute Step 2 of the dynamic analysis.

Select the plotted waveforms to display in the plot view using the Select Plots Form, illustrated in Figure 3.22. The Select Plots form is opened by clicking on the toolbar button . This form contains two tables: a table of available waveforms (right side), and a table with waveforms that have been selected for display (left side). Note that presently, all available waveforms are generated by Global Meters. Waveforms can be added and removed from the left table in several ways. Activate the **Global** and **Other** check-boxes, then click on the **Add Groups** button to generate the plot illustrated in Figure 3.23. This plot contains outputs of all global meters monitoring displacements forces and electric currents (i.e. all quantities other than stresses). Note that the **Add Groups** button puts all plots of the same quantity in the same plot frame, thus requiring only three frames to display a total of 15 plot traces. Alternatively the **Add All** button places each plot trace on a separate frame.

Next Activate the **Global** and **Stress** check-boxes, then click on the **Add Groups** button to generate the plot illustrated in Figure 3.24. This plot contains outputs of all tensile, compression and shear stresses. Note that the maximum stress is a compression stress and occurs on an element of ALUMINUM_PIPE_C type (i.e. a bus conductor).

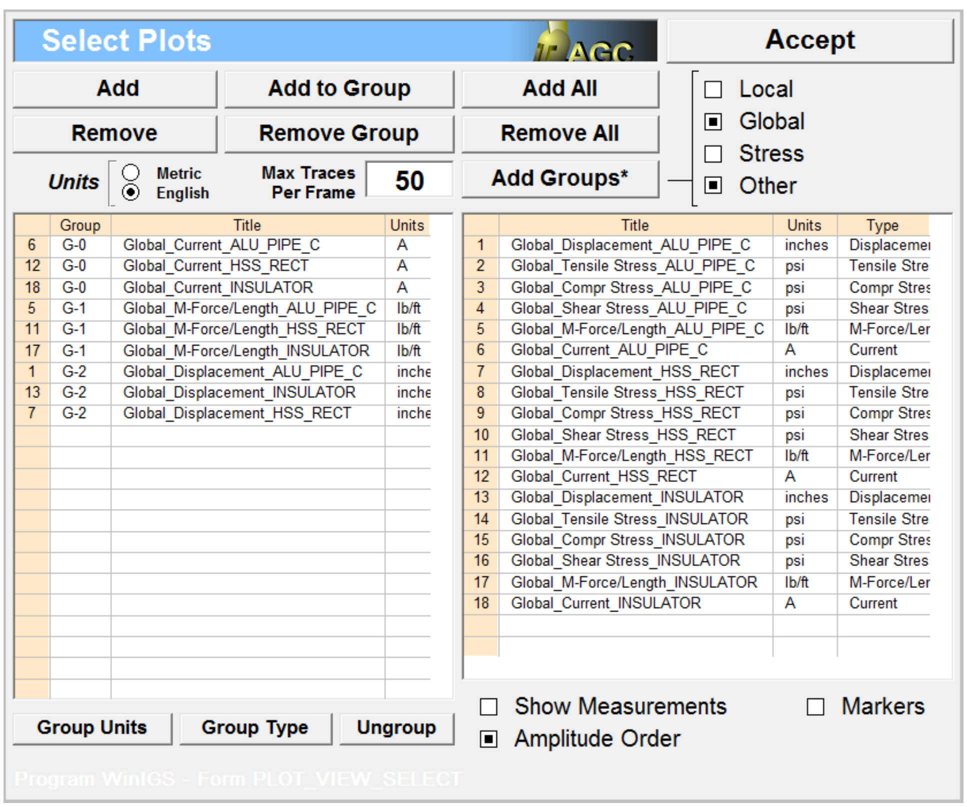


Figure 5.22: Plot Selection Form

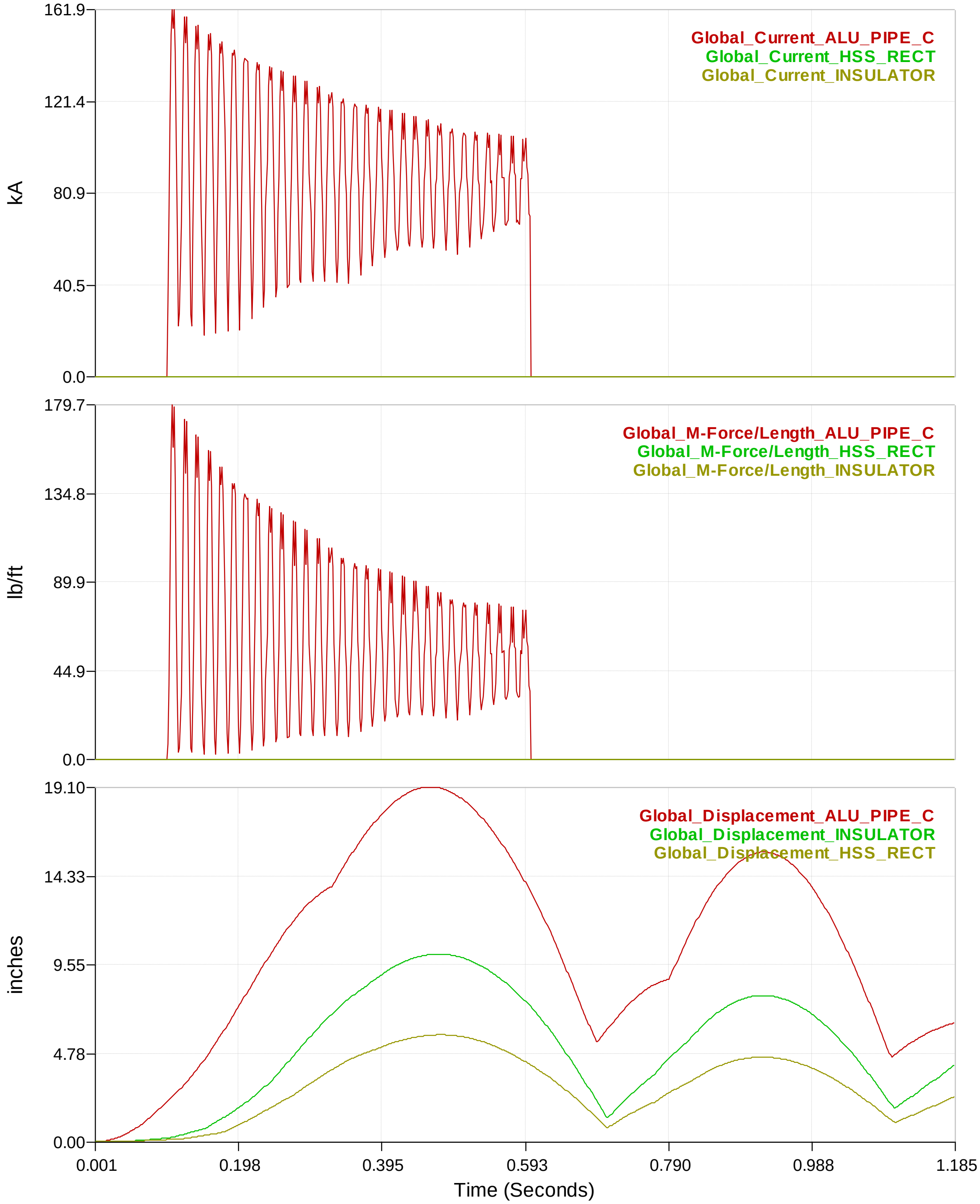


Figure 5.23: Plots Generated From Displacement, Force and Electric Current Global Meters

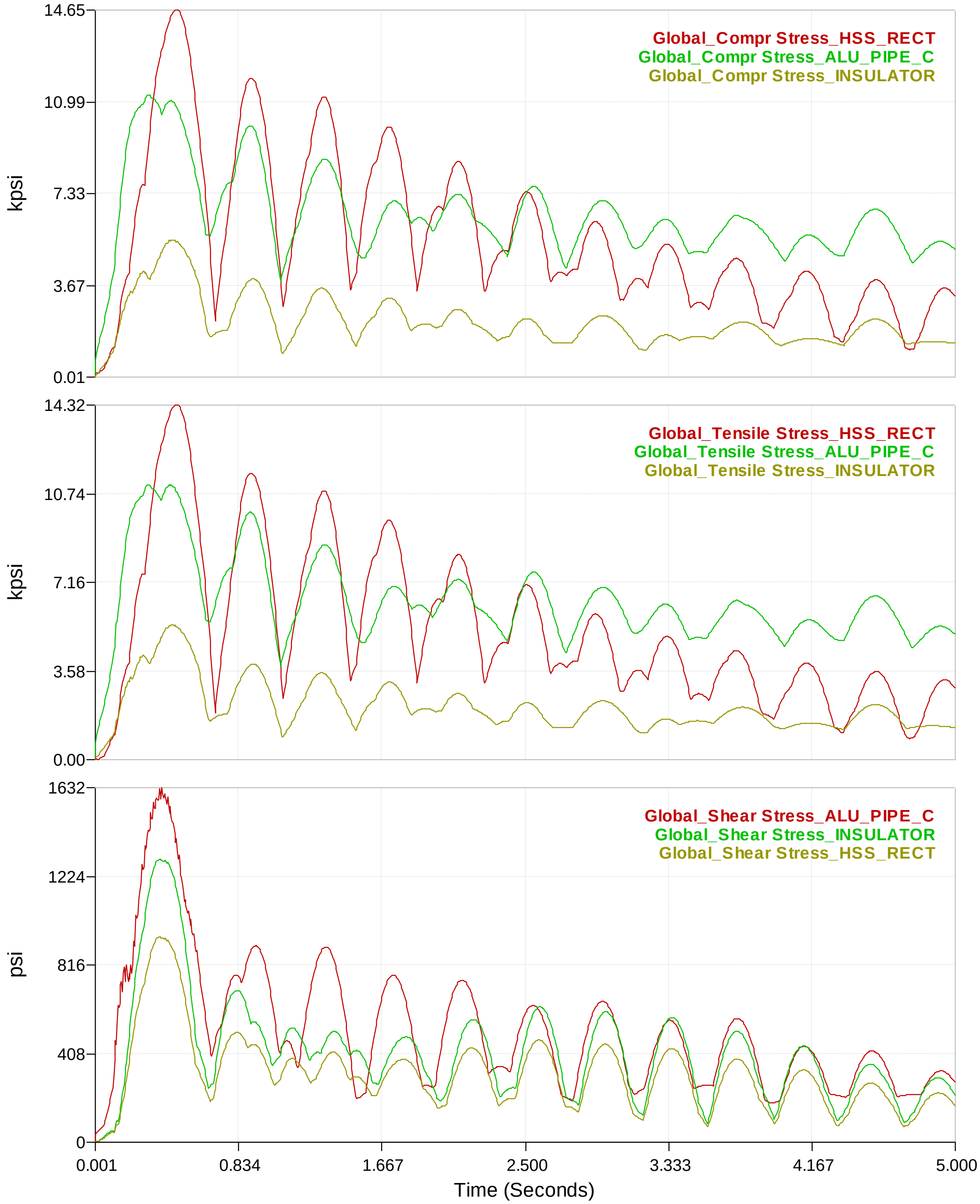
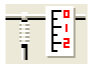


Figure 5.24: Plots Generated From Tensile Compression and Shear Stress Global Meters

5.5.4 SDA Step 3 – Stress and Displacement at Maximum Stress Points

In step 3 of the structural dynamic analysis, the stresses occurring at the points where maximum stresses occur are generated. This is achieved by automatically creating local meters at the locations identified by Global Meters during step 2.

Once the step 2 simulation is completed, open the meter element setup form, by clicking on the toolbar button . The form is illustrated in Figure 3.21. Click on the Create Meters button of the form to automatically create local meters at the locations identified by global meters. Note that a list of the created local meters appears in the Existing Meters table of the form. Furthermore meter symbols appear at the points monitored by the local meters on all views of the simulated system. The meter locations are numbered by an index number for easy identification. Figures 3.26 and 3.27 illustrate the meter locations on a 3-D rendered view and a top view respectively. Note the meter index numbers are visible in all wireframe views (Non Rendered Views).

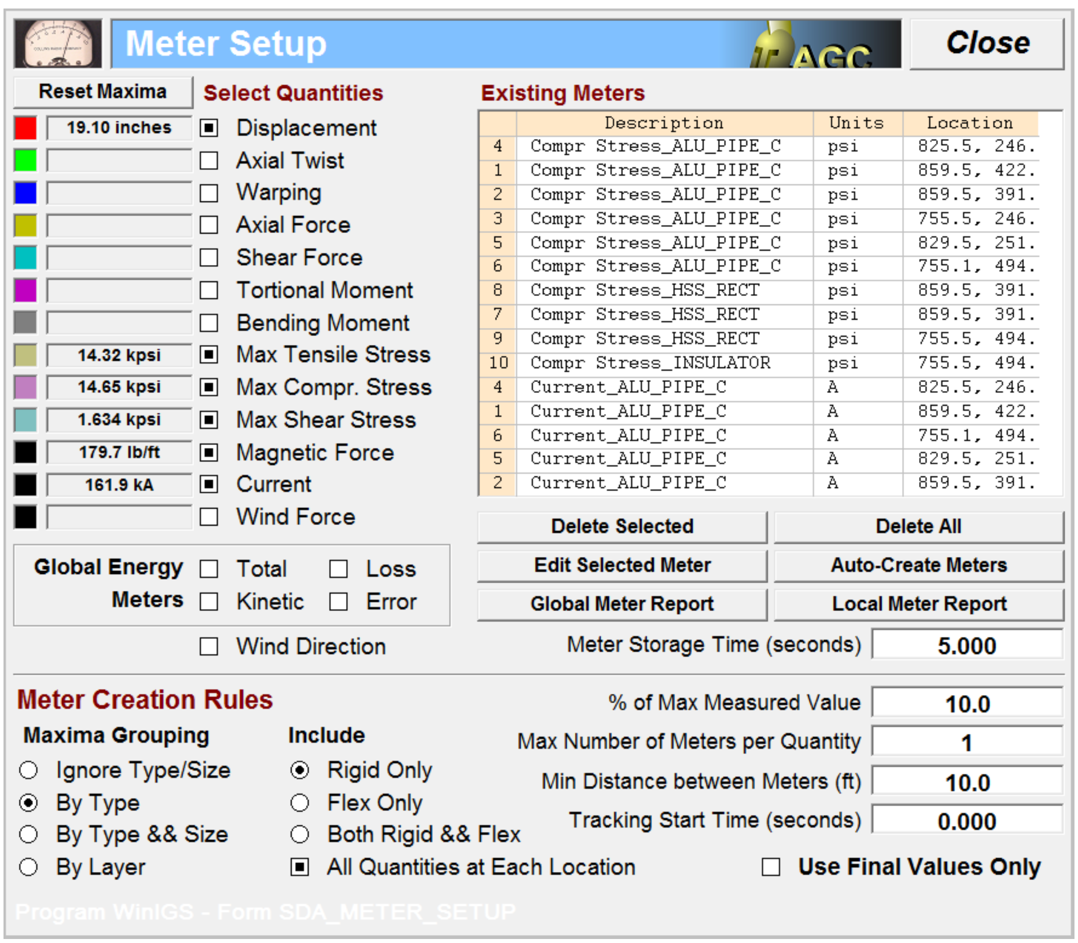


Figure 5.25: Meter Element Setup Form After Automatic Meter Creation

Set the stop time to 3.0 seconds and click on the START button in order to execute Step 3 of the dynamic analysis.

Once the simulation is completed, select the plotted waveforms to display in the plot view using the Select Plots Form. The Select Plots form is opened by clicking on the toolbar button: . Activate the "**Global"** and "**Other"** check-boxes, then click on the **Add Groups** button to generate the plot illustrated in Figure 3.28. This plot contains outputs of all local meters monitoring displacements forces and electric currents (i.e. all quantities other than stresses). Next, Activate the **Global** and **Stress** check-boxes, then click on the **Add Groups** button to generate the plot illustrated in Figure 3.29. This plot contains outputs of all tensile, compression and shear stress local meters. As expected given the global meter reports, the maximum stress is again a compression stress and occurs on an element of ALUMINUM_PIPE_C type. This maximum value is generated by a local stress meter at location on index #1 (red trace on middle plot frame of Figure 3.29). The location corresponding to index #1 is illustrated in Figure 3.30 - a wire-frame 3-D view of the simulated system.

A table summarizing the maximum stress values is generated using the **Local Meter Report** button of the Meter Element Setup Form (shown in Figure 3.25). An Example of a summary table is illustrated in Figure 3.31. Note that the displayed quantities can be selected using the controls located across the bottom of the form. Furthermore, the table can be sorted using any column as the sorting key. The sorting key column is selected by a left mouse button double click on the desired column.

Note that the summary table includes a **% Margin** value for each reported location point. The margin is defined as the difference between the maximum occurring tensile, compression, or shear stress and the corresponding material elastic limit, expressed as a percentage of the material elastic limit. A negative margin indicates that the material elastic limit has been exceeded.

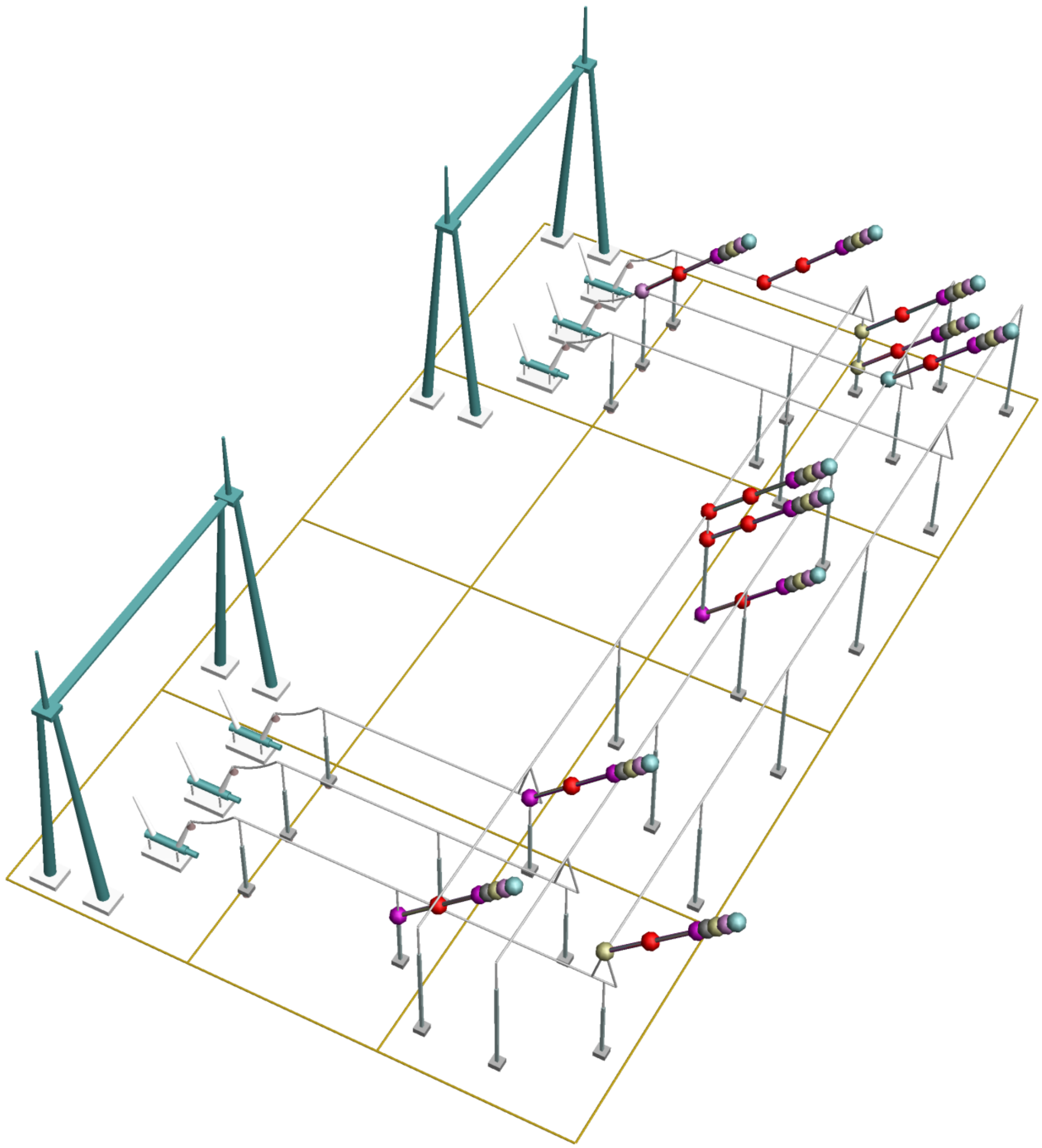


Figure 5.26: 3-D View showing Created Meter Elements

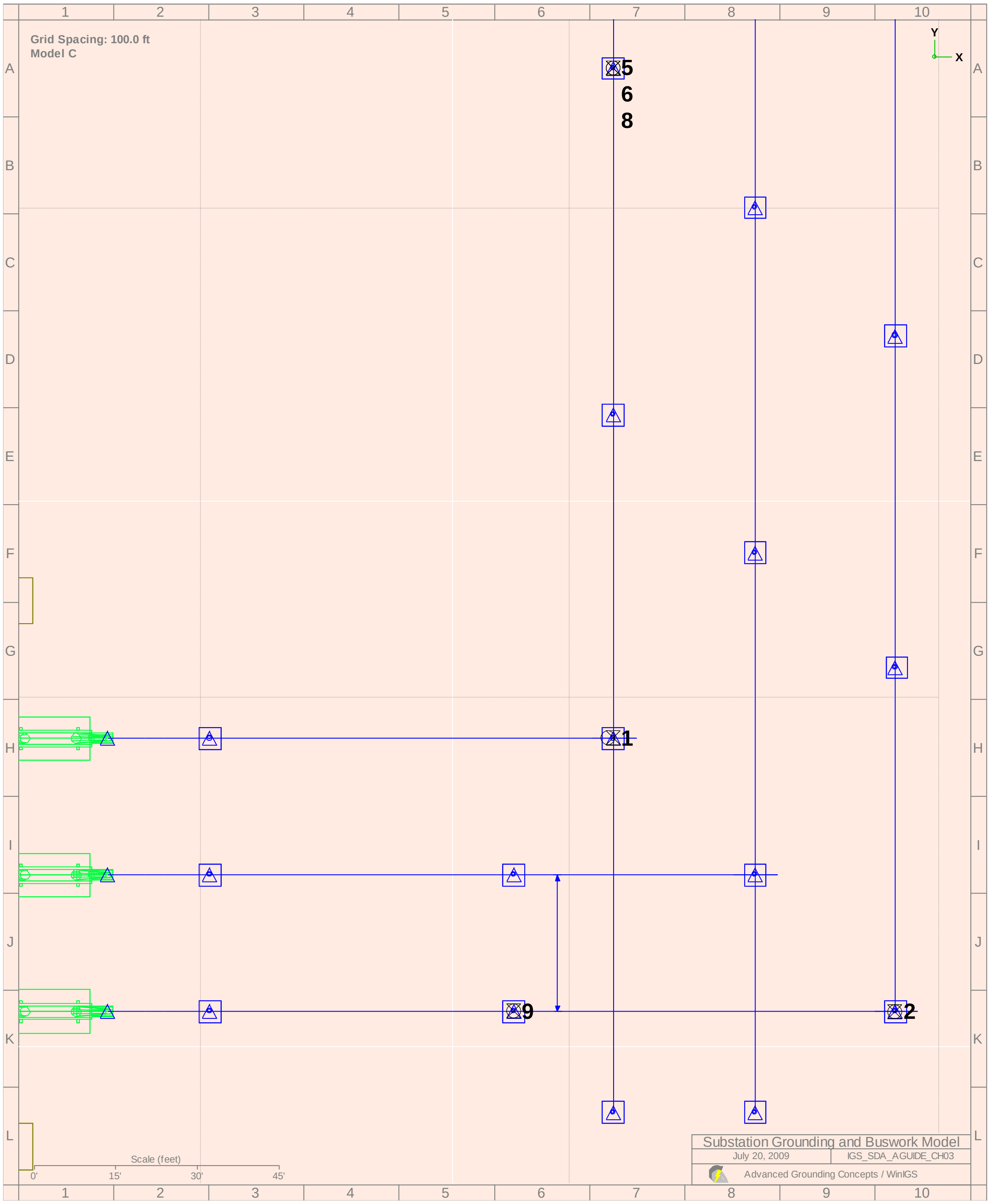


Figure 5.27: Partial Top View showing Created Meter Element Numbering

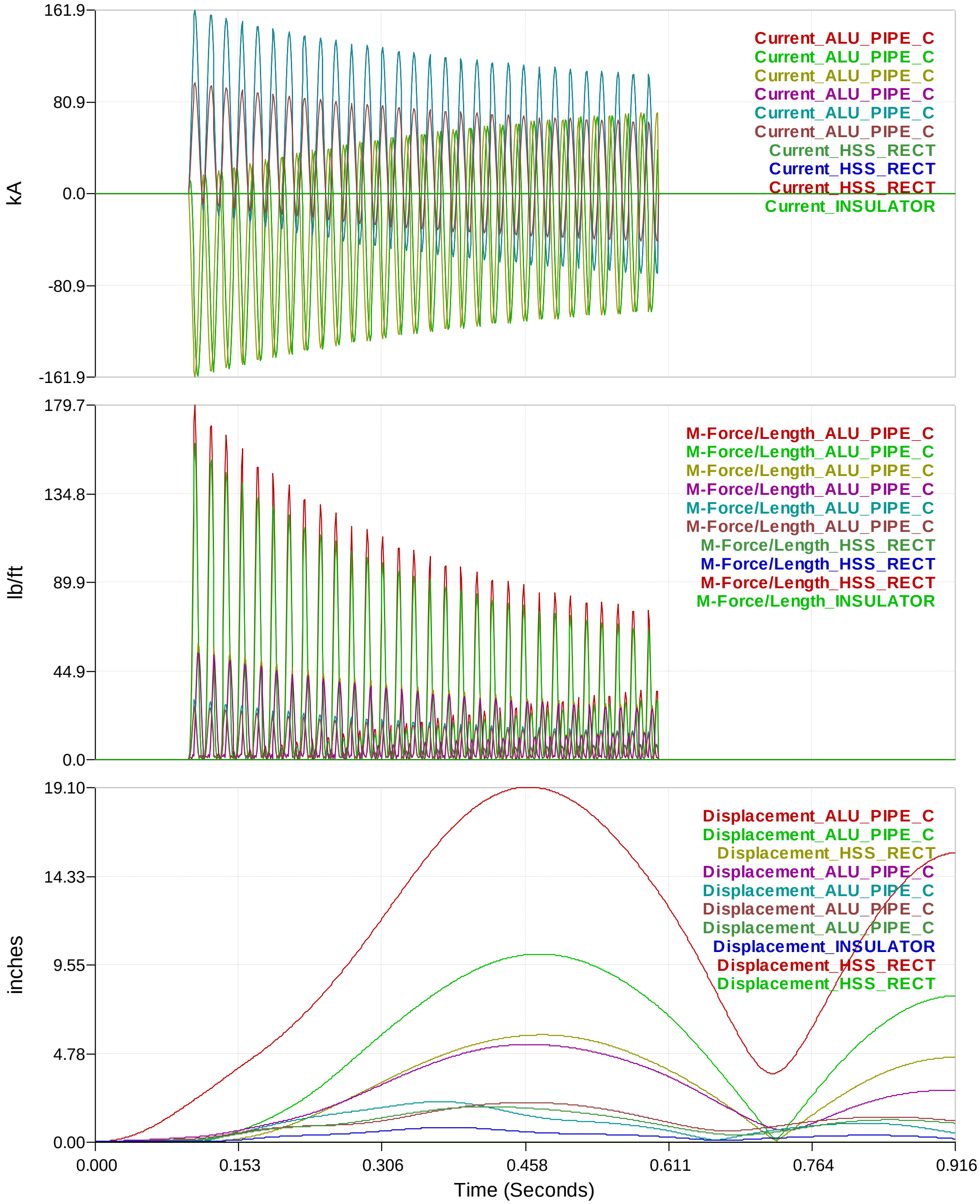


Figure 5.28: Plots Generated From Displacement, Force, and Electric Current Local Meters

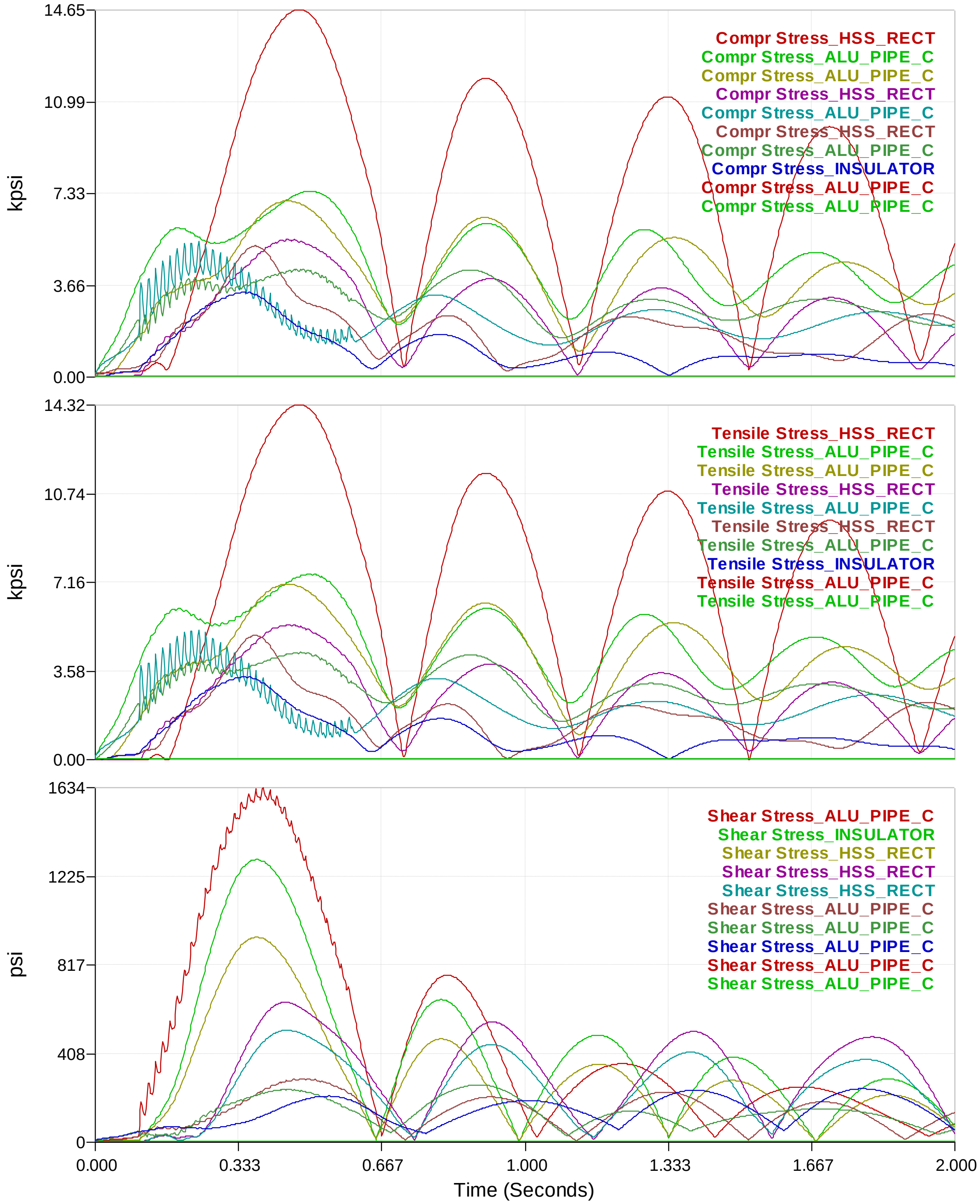


Figure 5.29: Plots Generated From Tensile Compression and Shear Stress Local Meters

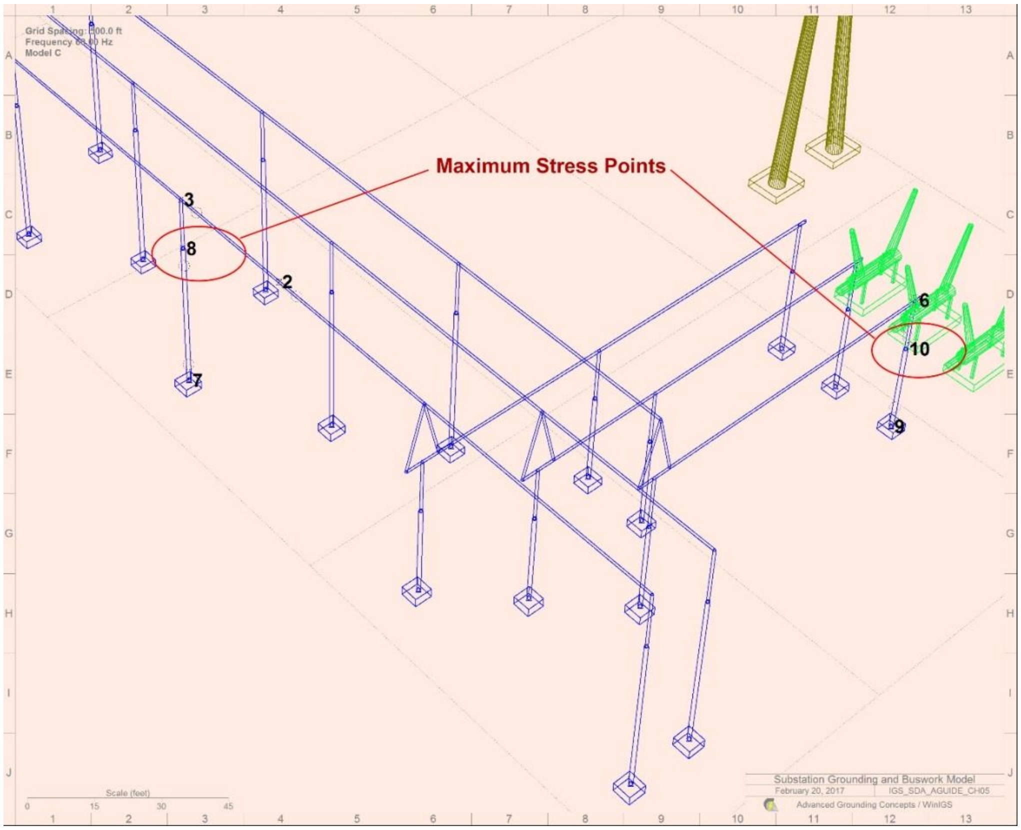


Figure 5.30: Location of Maximum Stress Point

SDA Local Meter Report 1 — CLOSE

Location	Element	Displacement	Tensile Stress	Compr Stress	Shear Stress	Allowable	Margin
8	HSS_RECT	5.758 inches	5.443 kpsi	5.485 kpsi	642.3 psi	2.817 kpsi	-96.02 %
10	INSULATOR	0.769 inches	3.339 kpsi	3.378 kpsi	1.301 kpsi	2.817 kpsi	-28.50 %
7	HSS_RECT	0.000 mils	14.32 kpsi	14.65 kpsi	513.8 psi	36.26 kpsi	59.56 %
3	ALU_PIPE_C	10.11 inches	7.484 kpsi	7.409 kpsi	244.1 psi	22.19 kpsi	66.26 %
2	ALU_PIPE_C	19.10 inches	7.078 kpsi	7.014 kpsi	291.1 psi	22.19 kpsi	68.08 %
1	ALU_PIPE_C	2.110 inches	5.254 kpsi	5.423 kpsi	1.634 kpsi	22.19 kpsi	74.48 %
5	ALU_PIPE_C	5.241 inches	4.328 kpsi	4.297 kpsi	262.5 psi	22.19 kpsi	80.46 %
9	HSS_RECT	0.000 mils	5.010 kpsi	5.220 kpsi	943.6 psi	36.26 kpsi	85.37 %
4	ALU_PIPE_C	1.880 inches	24.41 psi	23.28 psi	3.274 psi	22.19 kpsi	99.89 %
6	ALU_PIPE_C	2.155 inches	17.53 psi	16.69 psi	2.821 psi	22.19 kpsi	99.92 %

Sorted by Margin
Double Click on the Column To Sort

Display: ▣ Stress ☐ Force/Moment ▣ Deflection ☐ Coordinates

Units: ○ Metric ◉ English

Program WinIGS - Form SDA_METER_REPORT_1

Figure 5.31: Local Meter Report Summary Table

5.6 Discussion

Dynamic versus Static Analysis. This example illustrates the use of the Structural Dynamic Analysis tools of WinIGS to compute the stresses occurring on a rigid bus structure due to magnetic forces developing during an electrical fault. The analysis is based on a time domain simulation of the buswork dynamics. The bus conductors insulators supports, etc. are modeled using a finite element elastic beam model. This approach captures stress values occurring during oscillations, which typically exceed stresses occurring after the system reaches steady state. Thus, using dynamic analysis may identify structural failures that are missed by static analysis based techniques.

Global versus Local Meters. The stress, forces, and displacements at every point of the simulated system are monitored using global and local meters. At every time step of the simulation, each global meter scans all points of the simulated system and stores the maximum value of the quantity it monitors. It also stores the locations of *n* points where the largest *n* values occurred over the entire time duration of the simulation, where the number n is a user selected quantity. These locations are where Local Meters are automatically placed. Thus the time plot waveforms generated by global meters are not plots of a value at a single point, but an envelope curve that forms an upper bound of all the plots that can be generated by local meters at every point of the system. This approach avoids the huge storage requirement that would be necessary to store the time histories of all quantities at all points of the simulated system.

Excitation Options. In this example, the system was simulated under the influence of magnetic forces due to electrical fault currents (magnetic excitation). The effects of the structure component weights (gravity excitation) were also taken into account. Additional “excitation” options (not used in this example) include ice and wind loading, earthquakes, as well as user defined concentrated forces an moments.

Appendix A. Cantilever Beam

A.1 Introduction

This section presents the analysis of a single cantilever beam. The objective of this example is to provide a simple validation benchmark of the WinIGS structural dynamics solver. The WinIGS data files for the example system are provided under the study case name: IGS_SDA_AGUIDE_CH01.

The beam characteristics and support configuration are illustrated in Figure A.1. The steel beam has hollow square cross-section. One end of the beam is fixed (i.e. both position and rotation are fixed). The displacement of the free end of the beam is computed using analytic formulas and the results are compared to the WinIGS simulation results. The displacement is computed for two cases: (a) A 1000 lb vertical concentrated force is applied at the free end of the beam, while the weight of the beam is ignored, and (b) The weight of the beam is taken into account (but no concentrated force applied). Furthermore the first natural frequency of the beam is computed and compared to the beam frequency of oscillation during the dynamic simulation.

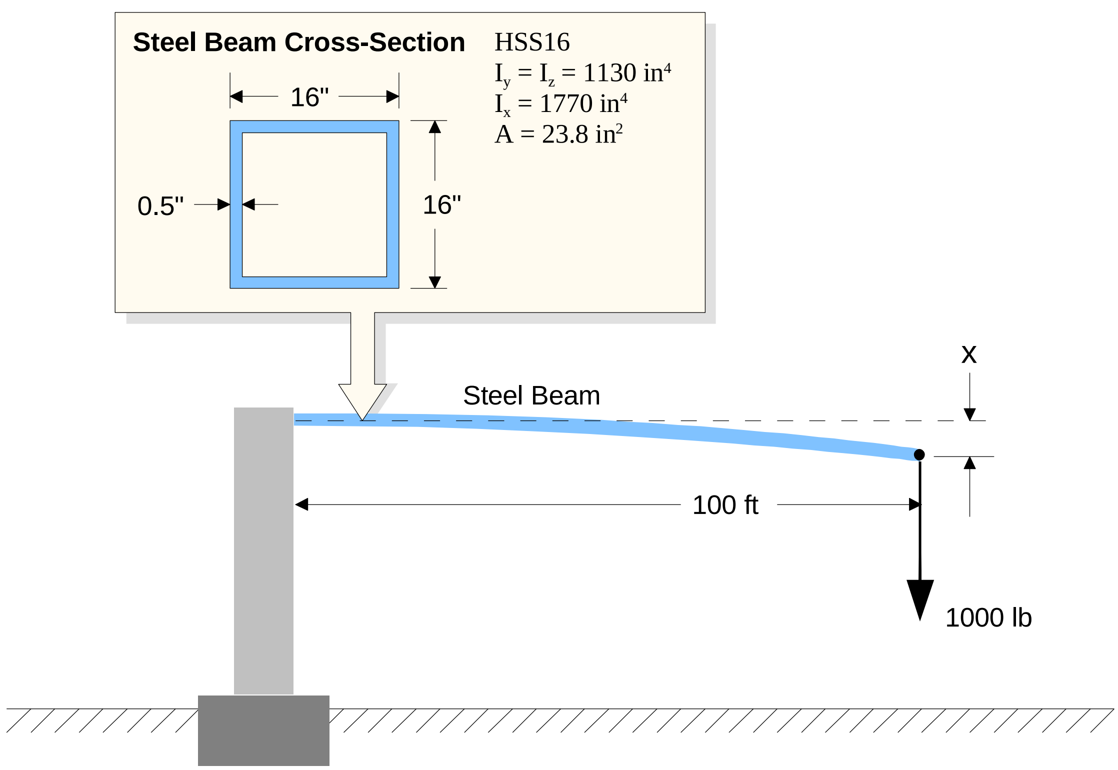
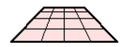


Figure A.1: Example System Configuration

A.2 Description of the System Model

In order to run this example, execute the program WinIGS and open the study case titled: IGS_SDA_AGUIDE_CH01. Use command **Open** of the **File** menu or click on the icon:  to open the existing study case data files. Note that the example study case data files are placed in the directory \IGS\DATAU during the WinIGS program installation.

Once the study case files are opened, the network editor window appears showing the system single line diagram, illustrated in Figure A.1. Note that this example case does not include an electrical network, and therefore the only model element defined is a "***Grounding System Element***" represented by the icon:  which contains the cantilever beam model. Double click on the ***Grounding System Element*** icon to open the grounding system editor and examine the cantilever beam model.

The grounding system editor is a 3-D modeling tool that can display three dimensional structures and allows graphical manipulation of the model elements. Specifically, the modeled system can be displayed in top view, side view, or perspective view. Use the following buttons to switch among these viewing modes (buttons are on the vertical toolbar on the left side of the WinIGS widow).

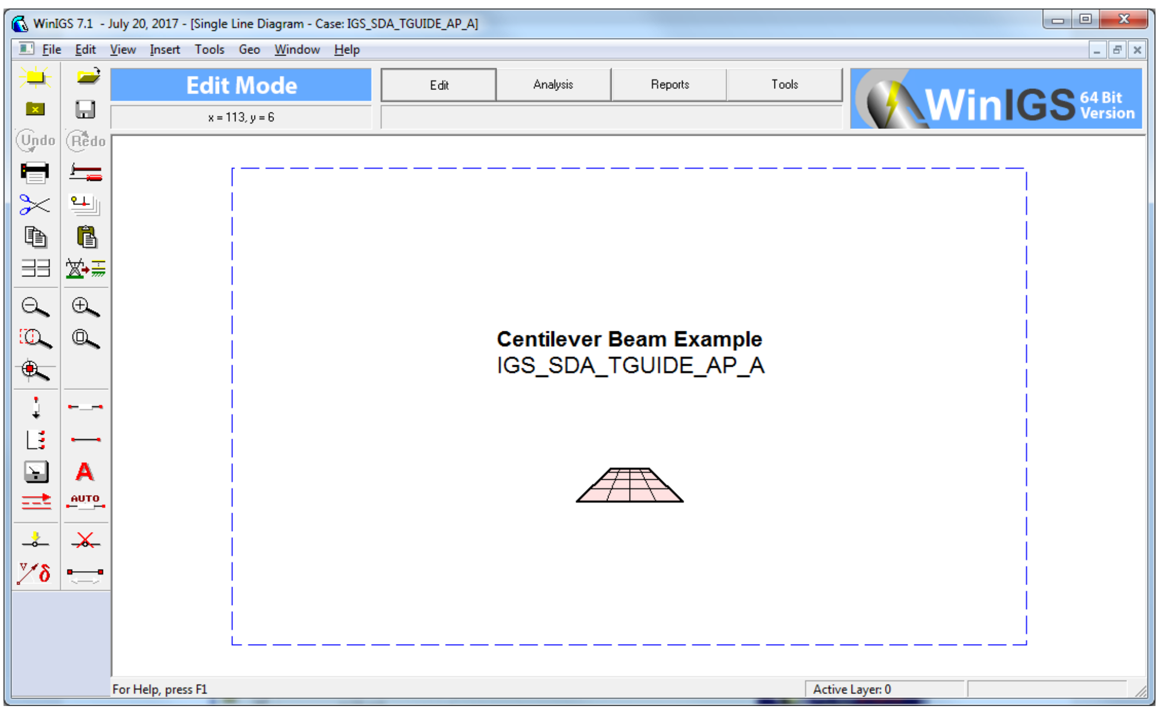
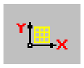
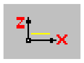
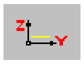
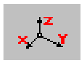


Figure A.2 Single Line Diagram of Example System IGS_SDA_AGUIDE_CH01

 Top view

 Side View

 Side View

 Perspective View5

 Rendered Perspective View

The default view is the top view of the modeled system (See Figure A.2). At any viewing mode you can zoom using the mouse wheel and pan by moving the mouse while holding down the mouse right button. In the perspective view mode, you can also rotate the view point by holding down both the keyboard Shift key and the right mouse button.

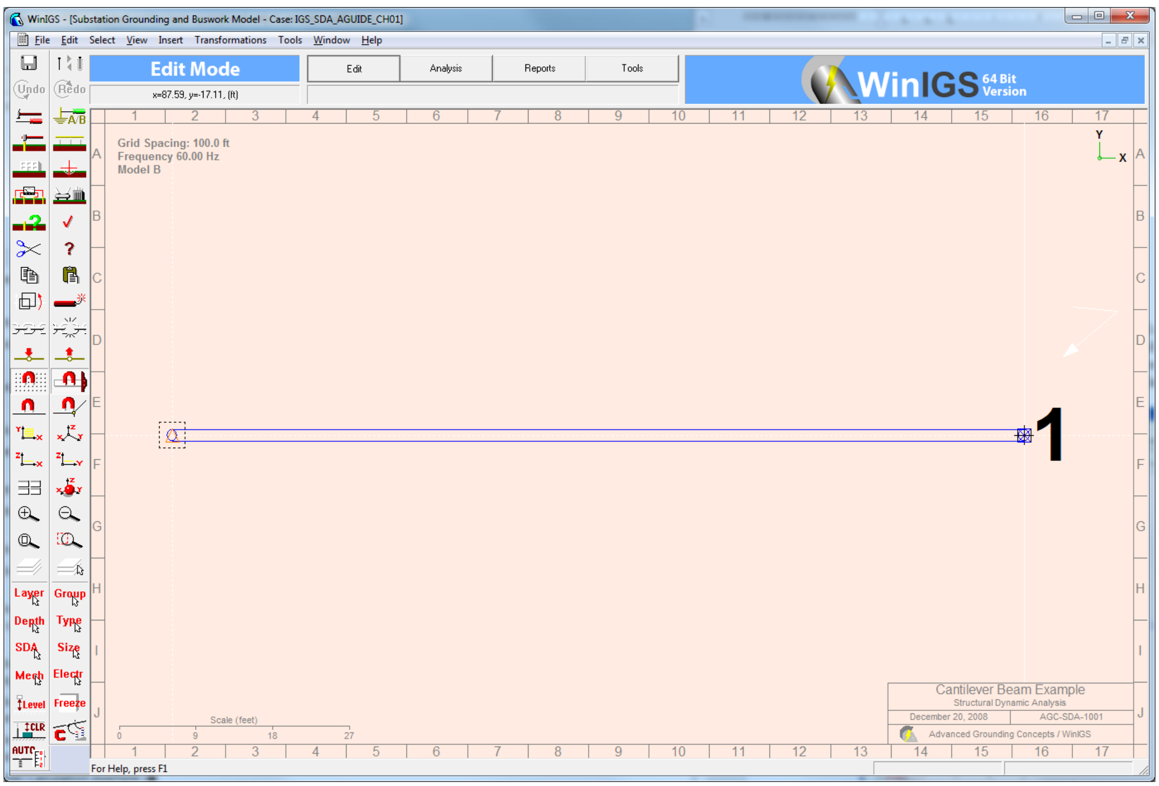


Figure A.3: Cantilever Beam Model – Top View

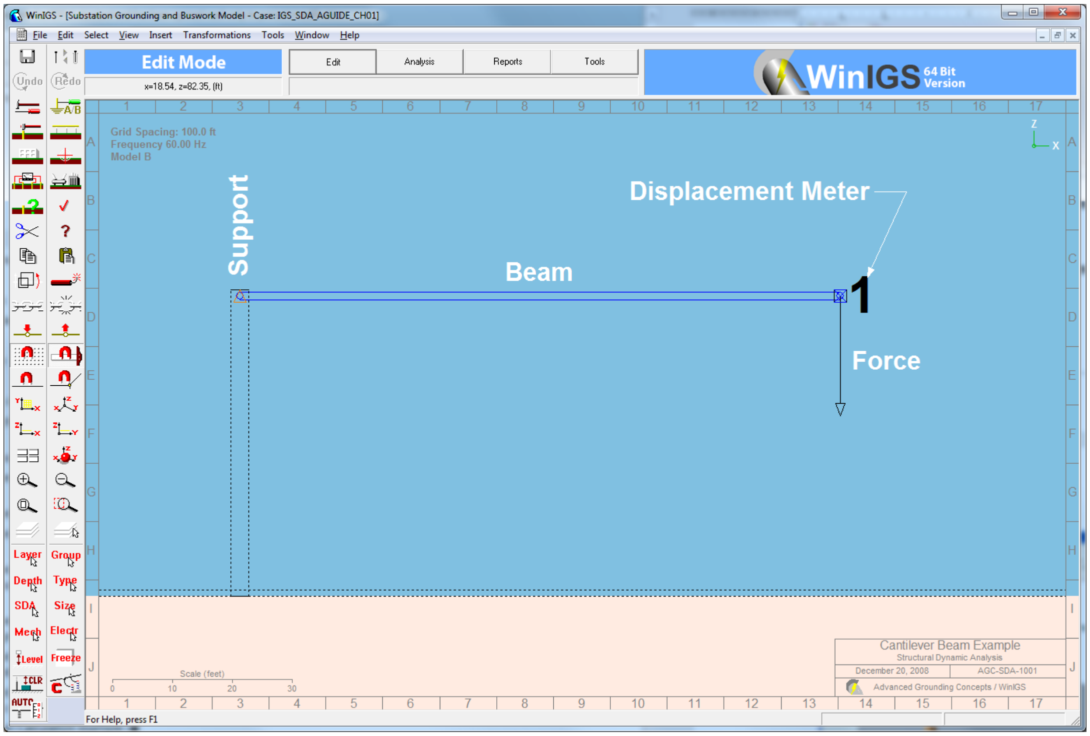


Figure A.4: Cantilever Beam Model – Side View

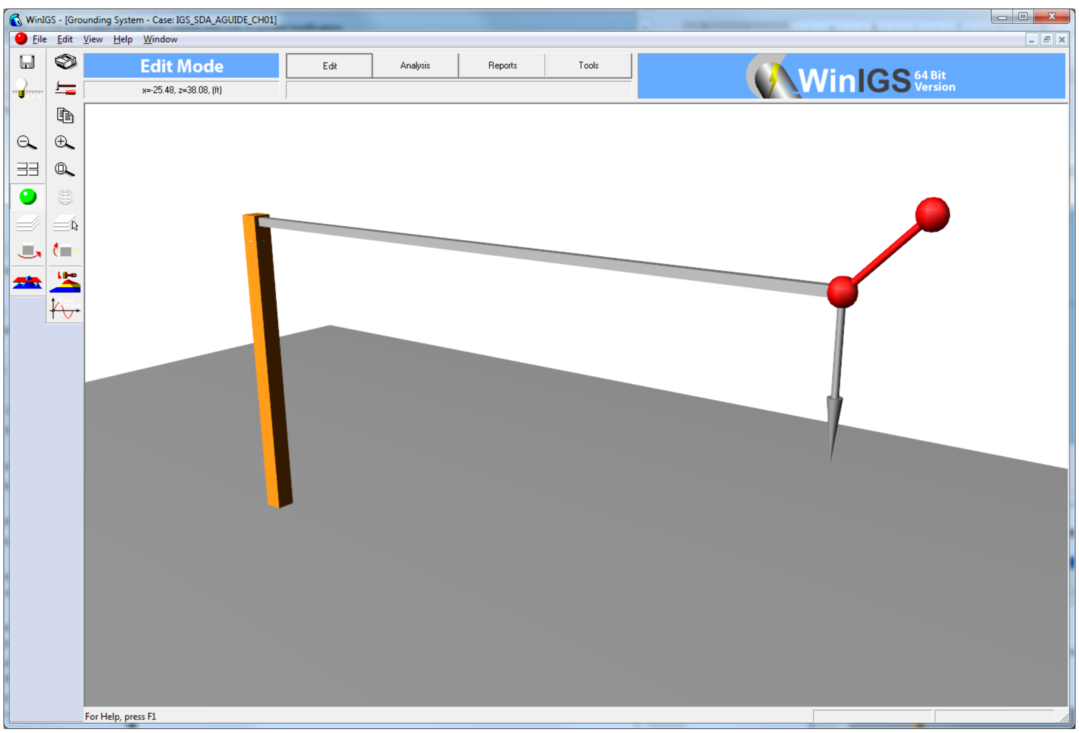


Figure A.5: Cantilever Beam Model – Rendered Perspective View

Next, examine the parameters of each model element. Double click on each element to open the corresponding parameter form. The parameters of the four main model elements of this example (beam, support element, force, and displacement meter) are discussed next.

Beam Element

The beam element parameters form is illustrated in Figure A.6. The form contains the following parameters:

Segment Coordinates. Coordinates are x, y, and z in feet for each endpoint. Z is the vertical direction with Z=0 corresponding to the earth surface. Note that coordinates can also be edited graphically by dragging the beam end points using the mouse.

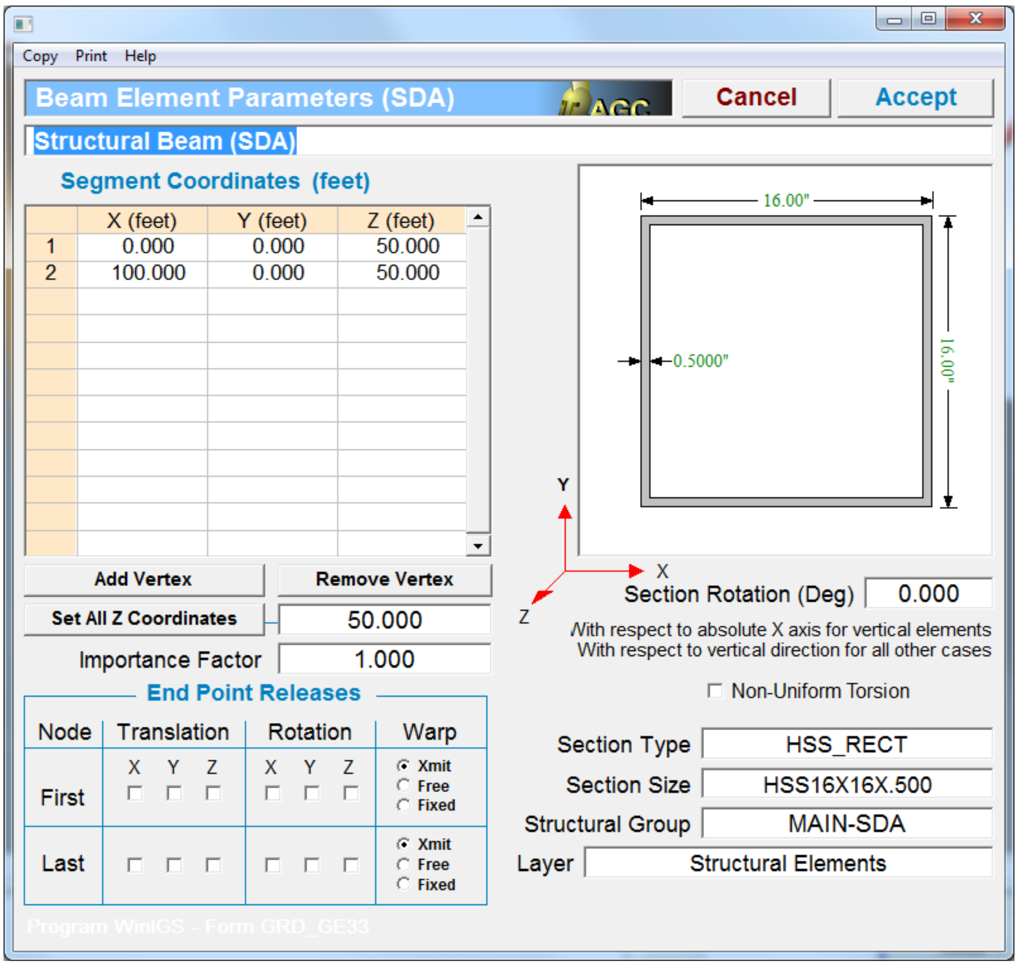


Figure A.6: Cantilever Beam Parameters

End Point Releases. These check-boxes isolate translational or rotational states at the beam ends in case the beam ends are connected (coincident with) other beam or support elements. Specifically, if all release check boxes are unchecked, the connection between this and other beams is taken to be rigid (such as a welding). Checking the X translation check box allows one beam to slide along the x direction. Checking the X rotational release allows the beam end to freely twist around the X axis (i.e. simulates hinge oriented along the x-axis), etc.

Section Rotation. Specifies the section orientation, defined by an angle of rotation about the main axis of the beam. The positive direction of rotation is determined by the right hand rule with respect to the major beam axis. The rotation angle is measured from the X-Axis if the beam main axis is vertical (z-directed), and from the vertical plane containing the beam-main axis if the beam major axis is not vertical. Figure A.7 illustrates the latter case. The rotation angle θ is measured from the segment AC. The segment AC is contained in the plane ABO, and also on the beam section plane.

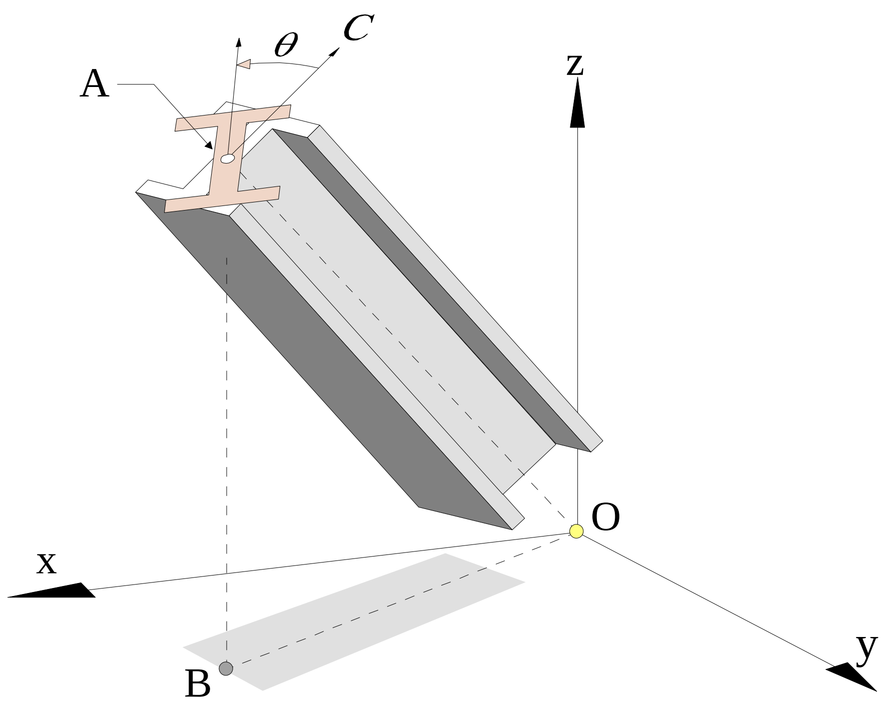


Figure A.7: Section Rotation

Type and Size. The beam type and size is selected from library tables. Click on either the Type or Size fields to access the beam library. Figure A.8 illustrates the beam library selection form. Note that the form displays the fundamental properties of the selected beam cross-section, namely, (a) the beam weight, (b) cross-sectional area, (c) beam material, and (d) the moments of inertia about the three major axes. Specifically J is the moment about the beam major axis (assumed to be X), and I_y, I_z are the moments about the horizontal and vertical axes of the beam cross-section.

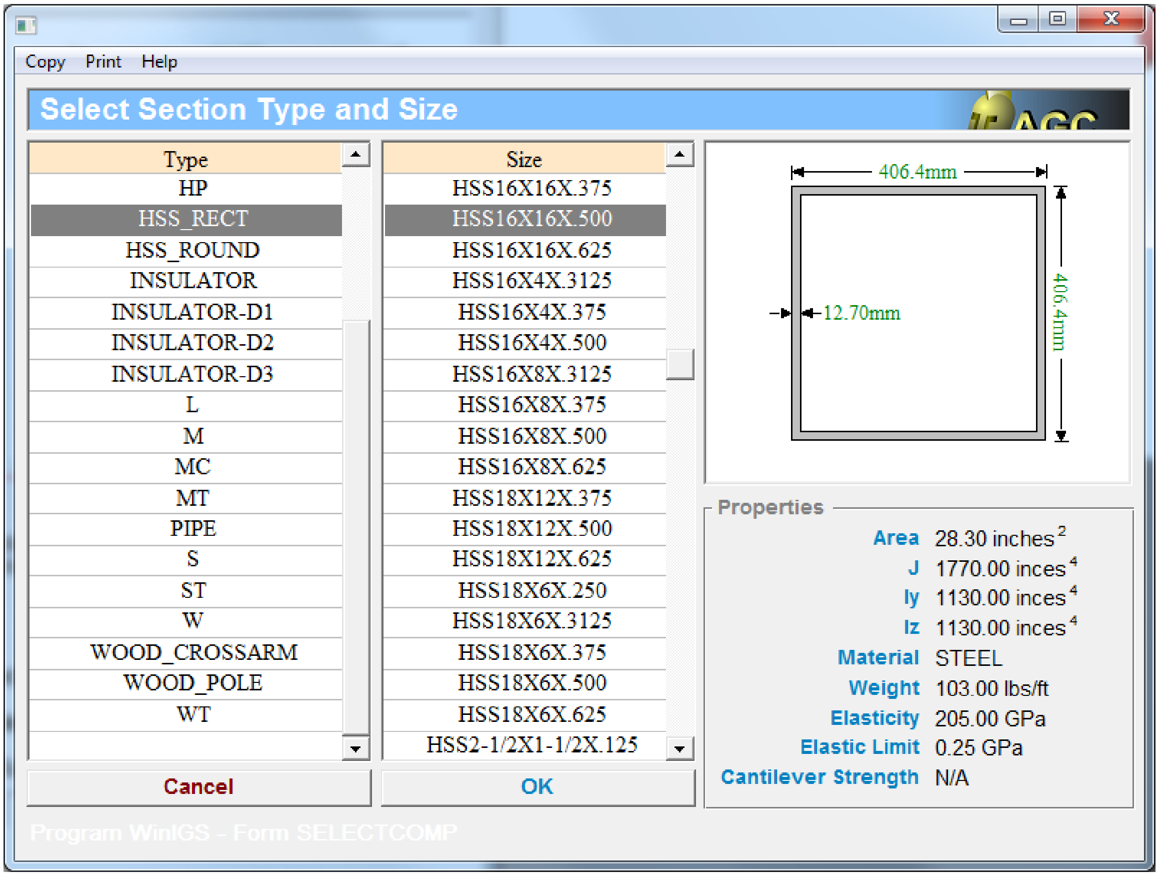


Figure A.8: Section Parameters

Cantilever Strength. This parameter is applicable only to components such as porcelain insulators for which the maximum allowable cantilever strength is specified by the manufacturer. The cantilever strength is defined as the maximum force perpendicular to the component main axis, that can be applied at the one end of the component while the other end is fixed. The force is specified in pounds.

Rigid Link. This checkbox changes the beam model to a simple rigid link model. A rigid link simply transfers moments and forces applied at its two ends without being deformed. Also, stress calculations are not performed on rigid links.

Structural Group. This is a user assigned name for the purpose of selecting groups of elements to include or exclude from the analysis. Structural analysis can be performed only the elements belonging to a user selected structural group (ignoring all other elements), or to all structural groups. On a complex system, performing structural analysis on one group at a time may be much faster than analyzing the entire system simultaneously. Of course, elements should only be placed in separate structural groups only if they are mechanically isolated from each other. A typical example containing mechanically isolated elements a rigid three phase bus, where the components of each phase can be placed in separate structural groups. See also section A.4.1 Mechanical Analysis Parameters Form.

Layer. The Layer property facilitates editing operations – it does not affect any analysis functions. For example, all elements in a specific layer can be hidden, editing operations can be prevented on selected layers, default colors can be assigned by layer, etc.

Support Element

The support element parameters form is illustrated in Figure A.9. The support element is represented by a triangular symbol. Support elements represent restriction of translation or rotational states on beam elements, such as foundations, hinges sliders etc. This example contains a single support located at the fixed end of the cantilever beam. The support element form contains the following parameters:

Center Coordinates. The x, y, and z, coordinates of the support element in feet. Z is the vertical direction with Z=0 corresponding to the earth surface. Note that coordinates can also be edited graphically using the mouse.

Structural Group. This is a user assigned name for the purpose of selecting groups of elements to include or exclude from the analysis. Structural analysis can be performed only the elements belonging to a user selected structural group (ignoring all other elements), or to all structural groups. See also See also section A.4.1 Mechanical Analysis Parameters Form.

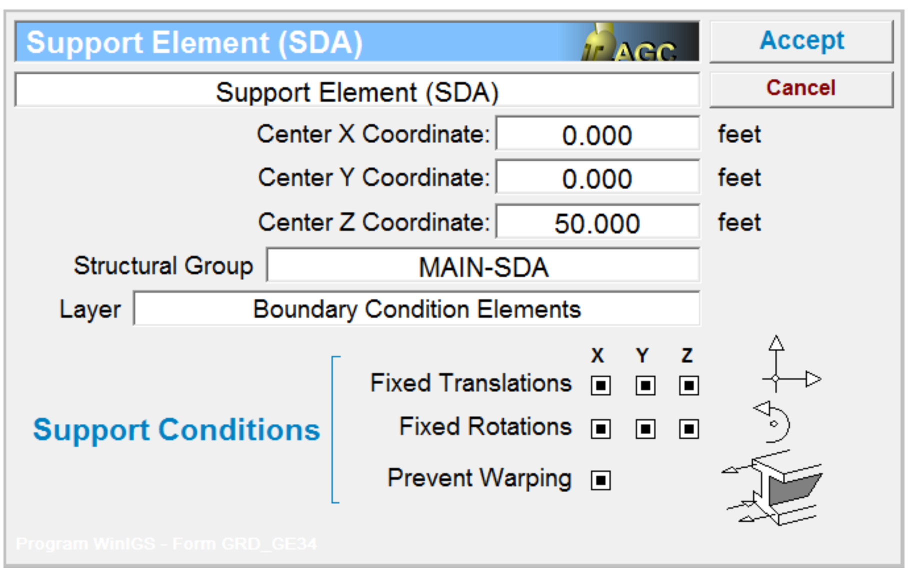


Figure A.9: Support Element Parameters

Layer. The Layer property facilitates editing operations – it does not affect any analysis functions. For example, all elements in a specific layer can be hidden, editing operations can be prevented on selected layers, default colors can be assigned by layer, etc.

Support Condition. These check-boxes activate the translational and rotational states to be fixed. Specifically, if all support condition check boxes are checked, the support is taken to be rigid (such as a fixed concrete foundation). Un-checking the X translation check box allows components connected to the support location to slide along the x direction. Un-checking the X rotational support condition allows the coincident elements to freely twist around the X axis (i.e. simulates hinge oriented along the x-axis), etc.

Note that for the purpose of solving the present problem, all support conditions (three translations and three rotations are fixed).

Mechanical Force Element

The mechanical force element parameters form is illustrated in Figure A.10. This element applies a concentrated force or moment at a selected location. The force element is represented by an arrow symbol indicating the direction of the applied force or the rotation axis of the applied moment. This example contains a single force element located at the free end of the cantilever beam. The force orientation is along the z-axis, pointing down.

The force or moment is a co-sinusoidal function with a user specified amplitude frequency and phase angle. Setting the phase to zero the force or moment assumes maximum value at time zero (cosine function). The mechanical force element form contains the following parameters:

Structural Group. This is a user assigned name for the purpose of selecting groups of elements to include or exclude from the analysis. Structural analysis can be performed only the elements belonging to a user selected structural group (ignoring all other elements), or to all structural groups.

Layer. The Layer property facilitates editing operations – it does not affect any analysis functions. For example, all elements in a specific layer can be hidden, editing operations can be prevented on selected layers, default colors can be assigned by layer, etc.

Amplitude. The peak value of the force on moment in pounds or pound-feet respectively.

Phase. The phase angle, in degrees, which is added to the argument of the cosine function that generates the force/moment time history.

Frequency. The frequency of the force/moment function in Hertz.

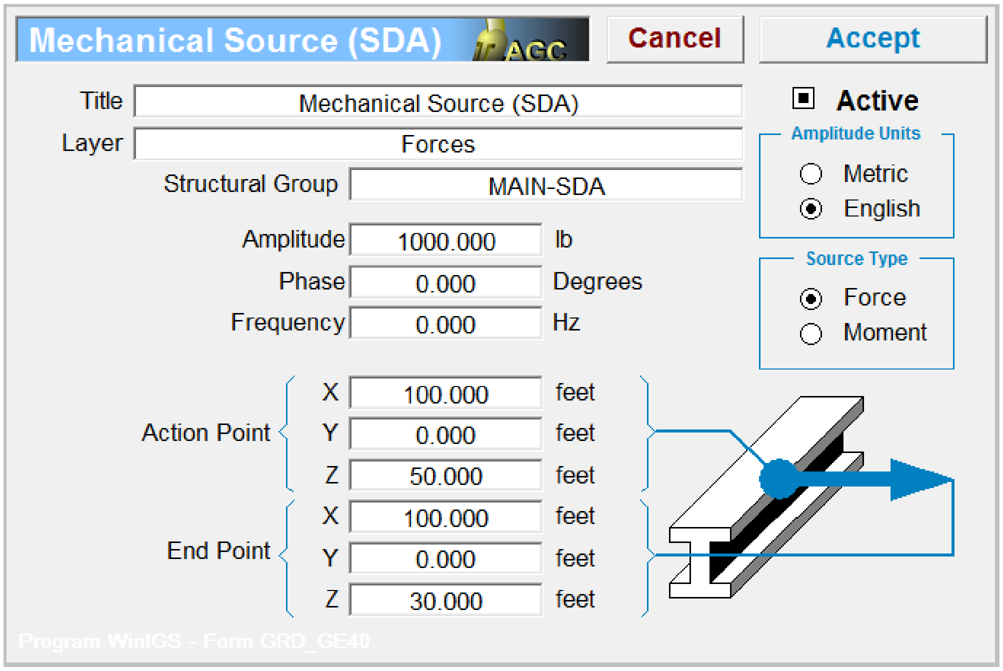


Figure A.10: Mechanical Source Parameters

Active. A check box that activates or de-actives the force or moment.

Force / Moment. Radio buttons for selection of force or moment application.

Action Point Coordinates. The x, y, and z, coordinates where the force or moment is applied in feet. Z is the vertical direction with Z=0 corresponding to the earth surface. Note that coordinates can also be edited graphically by dragging the source symbol using the mouse.

End Point Coordinates. The x, y, and z, coordinates of a second point, in feet, defining the direction of the applied force or moment. Note that the length of the force/moment symbol does not affect the magnitude of the applied force or moment.

Note that for the purpose of solving the present problem, a 1000 lb constant force is required. This is accomplished by setting the source amplitude to 1000 lb, and the frequency and phase to zero.

Meter Element

Meter elements allow the user to specify the quantities of interest to be reported during the system simulation. Meters can report displacements, forces, stresses, and electric current at user selected locations. Reports of meter quantities are presented in time plots, as well as tables of maximum values reached during the simulation. Note that meters can be manually created, or automatically generated during the simulation at locations where maximum values occur.

This example includes a displacement meter located at the free end of the cantilever beam. The meter is identified by the meter number (0 in this example) in the wire-frame views (top/side/3D) an by the red dumbbell symbol in the rendered-3D view. The meter element parameter form is illustrated in Figure A.11. The form contains the following parameters:

Title . A user assigned title identifying the meter element.

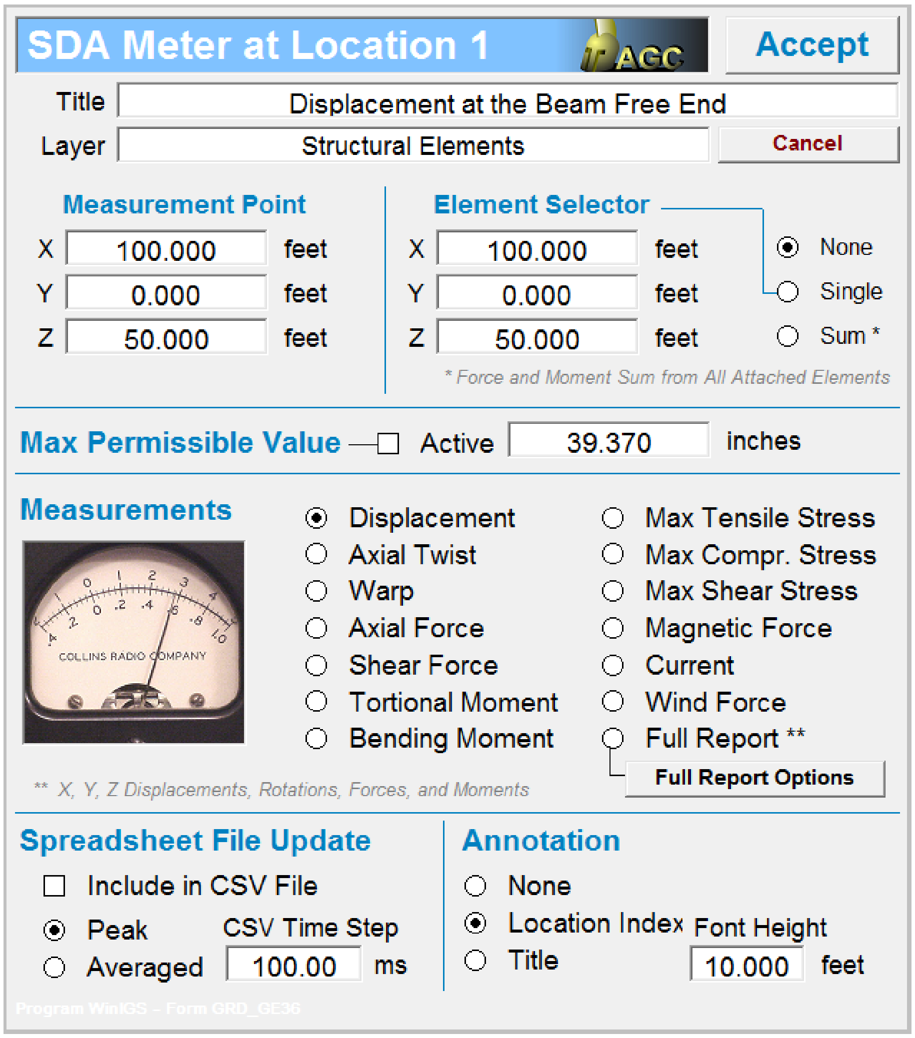


Figure A.11: Structural Dynamics Meter Parameters

Layer. The Layer property facilitates editing operations – it does not affect any analysis functions. For example, all elements in a specific layer can be hidden, editing operations can be prevented on selected layers, default colors can be assigned by layer, etc.

Measurement Point. The x, y, and z, coordinates (in feet) where the selected quantity is measured. Z is the vertical direction with Z=0 corresponding to the earth surface. Note that coordinates can also be edited graphically using the mouse.

Element Identifier. The x, y, and z, coordinates (in feet) of a second point. This point is by default coincident with the measurement point, and it is not displayed. For the purpose of displacement measurement, it is ignored. However, in force, moment, and stress measurements, it is useful in resolving the ambiguity that results if the measurement point is placed on the node connecting two or more beams. In this case the element identifier can be placed at a point along the length of the desired beam to be monitored. The <u>**Element Selector**</u> radio button titled "***Single***" must be selected to activate the element identifier symbol, and allow setting its position using the mouse.

Measurements. This control group contains 14 radio buttons which select the quantity to be monitored. Most of these radio button titles are self-explanatory. The last one titled Full Report, optionally reports all displacements, rotations, forces and moments applied at the measurement point on the selected beam element. These quantities are reported with respect to the global reference frame in Cartesian coordinates. A subset of these quantities can be selected using the button titled **Full Report Options**.

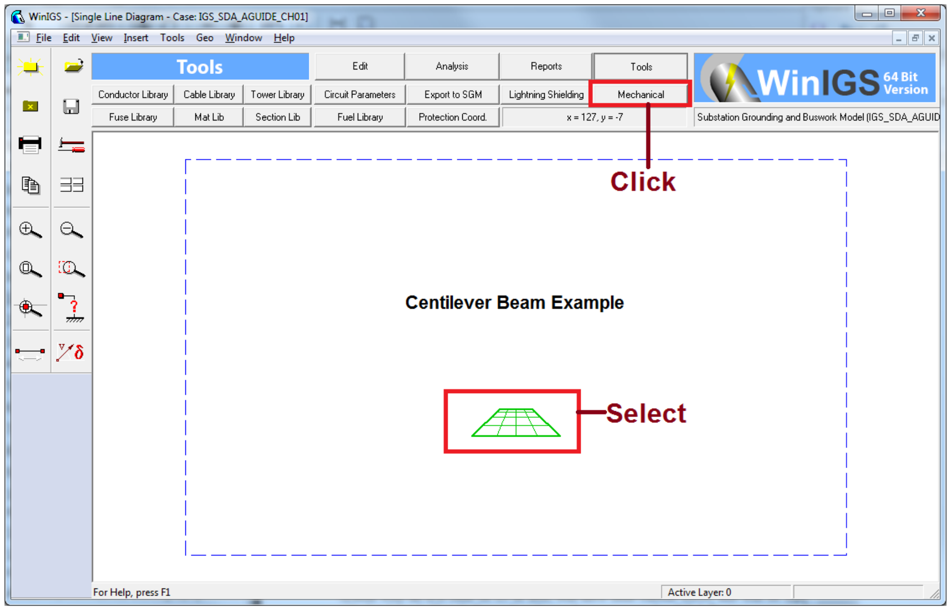
Spreadsheet File Output. These controls provide the capability of generating a standard CSV file containing waveform samples of the meter monitored quantities. If one or more meters have the checkbox titled "***Include in CSV file***" checked, a text file containing the output from all meters is generated. The **CSV time Step** parameter determines the sampling rate used for updating the CSV file. Note that the user value of the CSV Time step is adjusted to be an integer multiple of the actual simulation time step. The radio buttons titled **Peak** and **Average** determine how the values written in the CSV file are derived from the computed samples, if the CSV sampling rate is lower than the simulation time step. Specifically, the if the **Peak** button is checked, the values written in the CSV file are the maximum absolute value of the corresponding samples, while if the **Averaged** button is checked, the values written in the CSV file are computed by averaging the corresponding samples.

Annotation. Radio buttons for the selection of labels displayed next to the meter symbol. Two label options are available: (a) display of the meter location index number, (b) display the user assigned **Title**. The meter index is automatically assigned by the program based on the alphabetical ordering of the user entered meter titles. Note that if multiple meters are placed on the same location, the assigned location indices are identical.

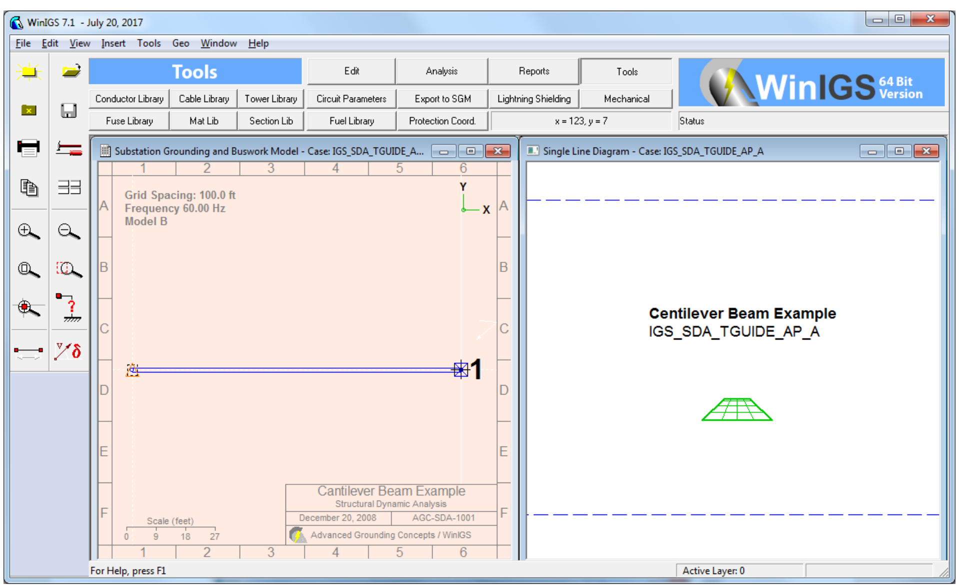
Font Height. Font height used for the meter index number shown in ground editor wire-frame view windows (in feet).

A.3 Running the SDA Simulation

In order to access the Structural Dynamic Analysis solver, click on the Tools Button (top row of WinIGS buttons), select the grounding system that contains the structural model of interest, and click on the Mechanical button (see Figure A.12a). The WinIGS mainframe window now contains the SDA controls, as illustrated in Figure A.12b

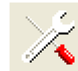


(a)



(b)

Figure A.12: Entering the Structural Dynamics Analysis Mode

Before executing the time domain simulation, click on the toolbar button  to open the simulation parameters window, illustrated in Figure A.13. (Also accessible via the Mechanical Analysis Parameters command of the Tools pull-down menu).

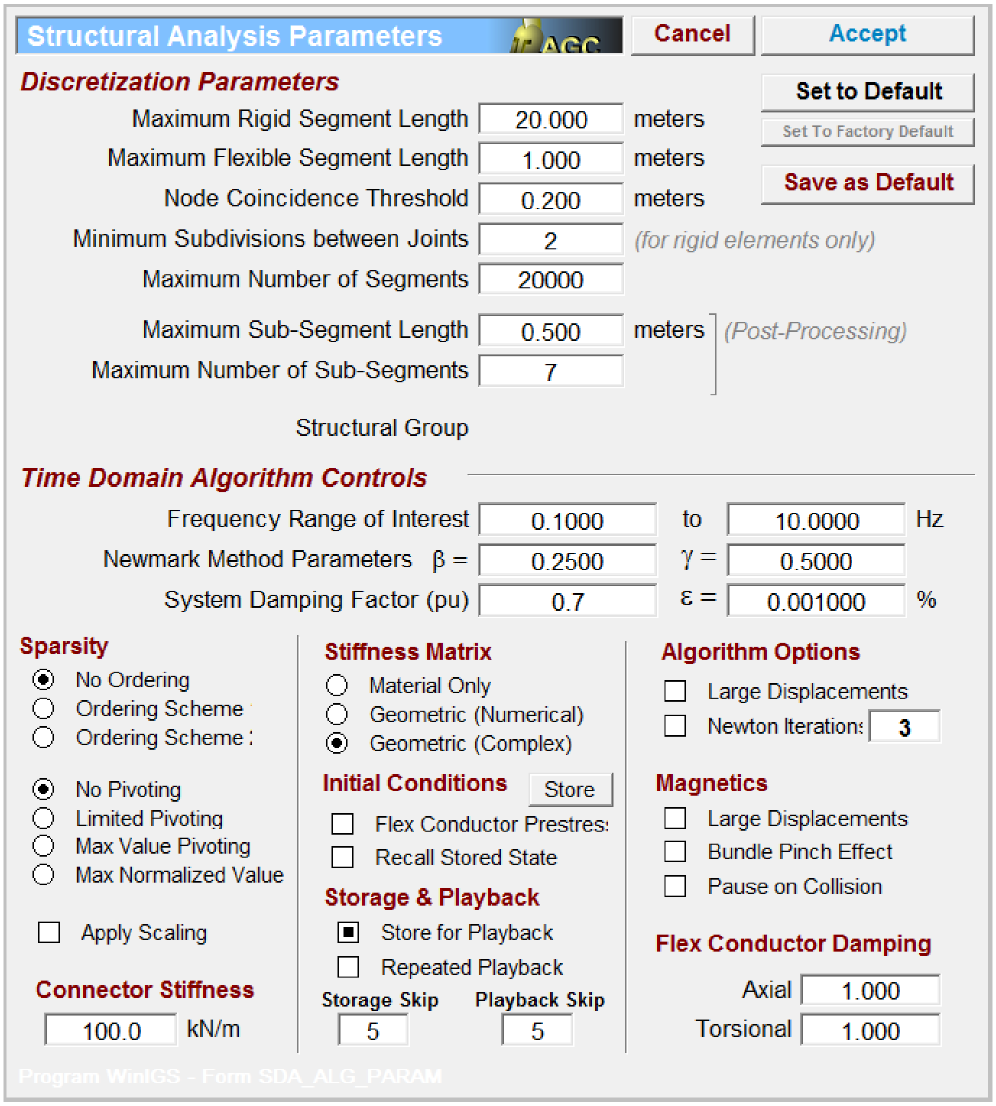


Figure A.13: Mechanical Analysis Parameters

A description of the simulation parameter from entries follows:

<u>**Discretization Parameters**</u> These parameters control the process of subdividing the components of the simulated structure into fundamental beam segments (finite elements), in order to provide an accurate solution. The default values of these parameters (listed in Figure A.11) are recommended for most applications.

Maximum Rigid Segment Length. The maximum length of a beam or rigid conductor element. Any element larger than this parameter is automatically subdivided into smaller elements.

Maximum Flexible Segment Length. The maximum length of a flexible conductor element. Any element larger than this parameter is automatically subdivided into smaller elements.

Node Coincidence Threshold. Any elements closer together than this value are considered to be joined together.

Minimum Subdivisions between Joints. The minimum number of beam elements between joints. Joints are the endpoints of all beams comprising the study case system as well as any crossing points among two or more beams.

Maximum Number of Segments. A desired limit on the number of beam elements after discretization. If this limit is exceeded, the solver automatically increases the maximum segment length. Note that in very large systems, this limit may be violated, since it may not be possible to reduce the final number of beam elements regardless of the Maximum Segment Length parameter**.**

Maximum Sub-segment Length. The beam element segments are subdivided into smaller segments (sub-segments) for the purpose of smoothly rendering the curved shapes that the beams assume under stress. This parameter, (along with the **Maximum Number of Sub-Segments**) controls the number and size of the generated sub-segments. Note that this segmentation only affects the accuracy of the rendered views of the system during the simulation. It does not otherwise affect the solution results.

Maximum Number of Sub-Segments. See description of *Maximum Sub-segment Length,* above.

Structural Group. This pull-down selection control allows selection of a subset of the modeled system to be used in the analysis. The selection is based in the structural dynamic analysis (SDA) group parameter specified for each component comprising the modeled system. The default SDA Group name is MAIN-SDA. The pull-down list box contains all the SDA group names occurring in the modeled system, plus the entry "*All SDA Groups*". If the *All SDA Groups* option is selected, all modeled elements are included in the analysis.

<u>Time Domain Algorithm Controls</u>

Frequency Range of Interest. The solver uses this frequency range (f_1, f_2) to determine the damping matrix for the analysis according the equation:

$$C = \frac{4\pi\zeta f_1 f_2}{f_1 + f_2} M + \frac{\zeta}{\pi(f_1 + f_2)} K$$

where: ζ is the user selected damping factor (See next parameter)
M is the system mass matrix
K is the system stiffness matrix
C is the system damping matrix

The frequency range of interest should be selected so that in includes the simulated system resonance and excitation frequencies.

System Damping Factor. Determines the rate at which transients are decaying. Recommended value for typical bus structures is 0.5. Note that in this example a higher value may be selected (such as 1.0) in order to reach the steady state solution within a short simulation time.

Newmark Method Parameters. These two parameters control the stability of the Newmark integration method. The recommended values are $\beta = 0.25$ and $\gamma = 0.5$.

Sparsity

Sparsity Coded Radio Button**.** Select this option to store the system matrices using sparsity coding techniques. This option is highly recommended since it provides high computational efficiency without any loss of accuracy. The computational efficiency benefits increase with the size of the simulated system.

Full Matrix Radio Button. Select this option to store the system matrices using full storage mode.

Ordering and Pivoting Method Radio Buttons. These options affect the efficiency of the system matrix sparsity coding. The optimal setting is dependent upon the simulated system topology. Experience with typical bus structures indicates that No Ordering and No Pivoting selections usually result in best performance.

Note that the algorithm analysis parameters dialog contains a number of additional controls (Stiffness Matrix, Initial Conditions, Connector Stiffness, Model Options, Magnetics, and Flex Conductor Damping) which do not affect the analysis of this example system. These controls are discussed is later sections.

Connector Stiffness. Affects the accuracy of representation of advanced connectors such as sliders and hinges. A high value (at or above 100 kN/m) is recommended.

Stiffness Matrix. Radio buttons select the method of computation of the element tangent stiffness matrix. Applicable only if the ***Large Displacement*** option is also selected. The Geometric Complex option is recommended.

Connector Stiffness. Affects the accuracy of representation of advanced connectors such as sliders and hinges. A high value of 100 kN/m is recommended. This value may be increased if connector miss-tracking is observed.

Initial Conditions

Store Present State. This button allows the user to store the present state of the simulated system to be used as an initial condition for a subsequent simulation. This feature is useful in simulating low damping systems for which steady state conditions under some constant force, such as due to gravity, take a long time to reach. In such cases the steady state solution is first computed and stored by simulation the system with the system damping factor set to a high value (e.g. 0.7). Subsequent simulations are executed with the actual system damping and starting from the pre-computed and stored steady state, by checking the **Recall Stored State** check box.

Recall Stored State. Check this box to assume a stored system state as the initial conditions of the simulation. Note that the system state must have previously been computed and stored using the **Store Present State** button (See also **Store Present State** topic above**).**

Storage and Playback

Store for Playback. Set this check box in order to enable solution storage for playback. The playback option is useful in the analysis of large systems, as playback can be carried out much faster than simulation. After a simulation is completed with the play back storage activated, any number of meters can be added or edited, and new results are obtained using the playback option.

Note that playback data are stored in two files, with the same file path as the case data file path ending with .PB1 and .PB2 respectively. The .PB2 file is usually very large since it contains the beam and connector element state variables for each simulation iteration.

Repeated Playback. This check box causes solution playback in an endless loop.

Storage Skip. Sets the number or iterations skipped before the beam and connector states are stored.

Playback Skip. Sets the number or beam and connector state records are skipped before animation views are updated. Increasing Playback Skip usually increases playback speed.

Algorithm Options

Large Displacements. This check box enables the corotational analysis method. If the structure under study contains large spans of flexible conductors, or in general large displacements may occur, then it is recommended to activate the **Large Displacements** check box.

Newton Iterations. This check box along with the numeric entry box sets the number of sub-iterations used to improve the solution accuracy. In most cases 2 or 3 Newton iterations are enough for achieving good accuracy during transients. Not that the steady state solution is not affected by these controls.

Magnetics

When analyzing magnetic forces on flexible conductors exhibiting large displacements, the **Large Displacements** check box, located under the **Magnetics** heading should also be checked.

Flex Conductor Damping

Axial and Torsional. These Flex Conductor Damping controls should normally be set to 1.0. They may be increased to values slightly above 1 (for example the value of 1.1) in cases where solution is unstable.

General Not on Solution Stability. While the linear analysis option (Large Displacements check box unchecked) is always stable, the nonlinear analysis algorithm invoked by the **Large Displacements** option may become unstable under second conditions. Stability can be achieved by the following settings: (a) increasing the system damping factor, (b) reducing the simulation time step. (c) Decreasing the maximum segmentation lengths (Maximum Rigid & Flexible Segment Length fields). In cases containing flexible conductors, the Flex Conductor Damping controls (Axial and Torsional) may be also increased to values slightly above 1 (for example the value of 1.1).

Simulation Controls While in SDA mode, the main WinIGS toolbar, illustrated in Figure A.14, contains controls that allow starting pausing and stopping the simulation, as well as selection of the simulation timing parameters (time step, duration of simulation, simulation speed, excitation options etc). These control are described below.

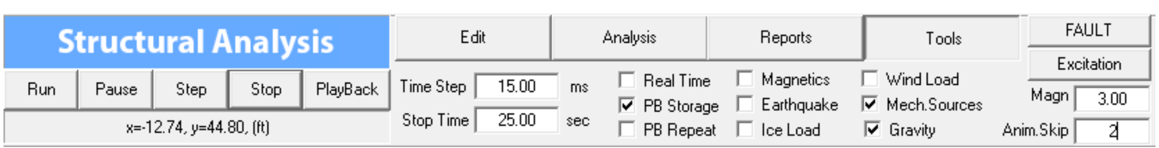


Figure A.14: Main SDA Mode Toolbar

Run/Pause/Step/Stop/Playback Buttons. Use these buttons to control the simulation or playback of the dynamic analysis of the modeled system.

Time Step. The time increment between successive time instants for which the system state is computed. For accurate results the time step should be a magnitude of order smaller than the highest frequency of interest. In the present example a time step of 15 milliseconds is used.

Stop Time. Defines the duration of the simulation. The simulation automatically stops when the simulated system time reaches this value. NOTE: Setting the stop time to zero the analysis continues indefinitely, until the STOP button is clicked. In the present example stop time is set to 25 seconds. With system damping factor set to 0.7, and after 25 seconds from the simulation initiation, the oscillations decay enough to obtain an accurate reading of the beam deflection at steady state.

Real Time Check Box. The simulation is executed in real time if possible. The ability to execute a simulation at or faster than real time depends on the system size and the selection of time step.

PB Storage Check Box. Enables the storage of the system state during the dynamic simulation, thus enabling the ***Play Back*** function

PB Repeat Check Box. Check this box to enable repeated playback (endless loop).

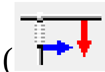
Magn. This entry filed sets the animation magnification factor. During the analysis the animated rendered 3-D view displays an animated view of the simulated system displacement. The displacement is exaggerated by this factor. This feature enables the effective visualization of the simulated system deformation and movement in cases where the actual displacements are too small to be visible. Note that this control is active only for small displacement analysis (i.e. when the Large Displacement check box is unchecked).

Anim. Skip. The entry field sets the rate at which the rendered animation views are refreshed. Specifically, setting this field to the value N, results in refreshing the animated views every N iterations of the simulation algorithm. Setting the skip factor to a higher value increases the simulation speed by reducing the computational effort expended in graphics.

Excitation Selection Controls

WinIGS Structural Dynamic Analysis provides six excitation force options, namely:

1. Magnetics: Forces due to electric currents circulation in buswork and earth.
2. Earthquake: Excitation due to earth motion during an earthquake.
3. Ice Load: Forces due to ice accumulation on buswork.
4. Wind Load: Forces due to wind on buswork.
5. Mech. Sources: Concentrated forces or moments at user selected points.
6. Gravity: Forces due to gravity .

These six check boxes located in the main SDA toolbar activate these excitation options. Additional parameters of each excitation option are accessible via the excitation parameters form, illustrated in Figure A.15. This form is opened by clicking on the **Excitation** button () of the main SDA toolbar.

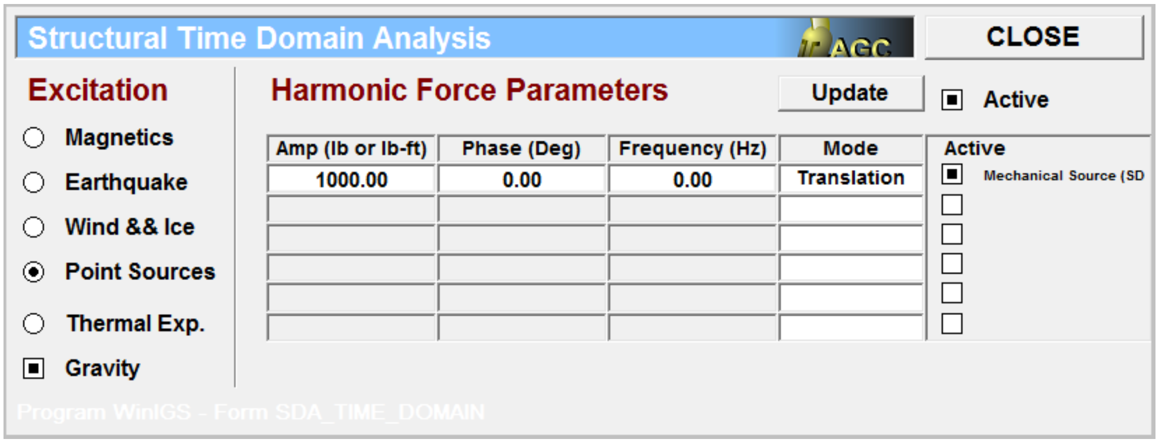


Figure A.15: Excitation Parameters Form

In the present example, the excitation is provided by a single concentrated force source (Point Source). The parameters of the point source can be examined and modified using this form. Note that the source parameters can be also edited by double clicking on the point source symbol in any of the 2-D or 3-D view windows displaying the simulated system. Point sources are co-sinusoidal functions with user defined amplitude, frequency and phase angle. In the present example, a constant force is applied by setting the phase and frequency to zero.

Dynamic Analysis. After inspection of the system parameters click on the START button of the main SDA toolbar to execute the dynamic analysis. Upon initiation of the analysis, a 3-D rendered window and a plot window are automatically opened. The 3-D window, illustrated in Figure A.16, displays an animated 3-D view of the simulated system. The plot window, illustrated in Figure A.17, displays plots of user selected quantities as a function of time. During the analysis execution, both animated rendered windows and plot windows are continuously updated providing effective visualizations of the simulation results.

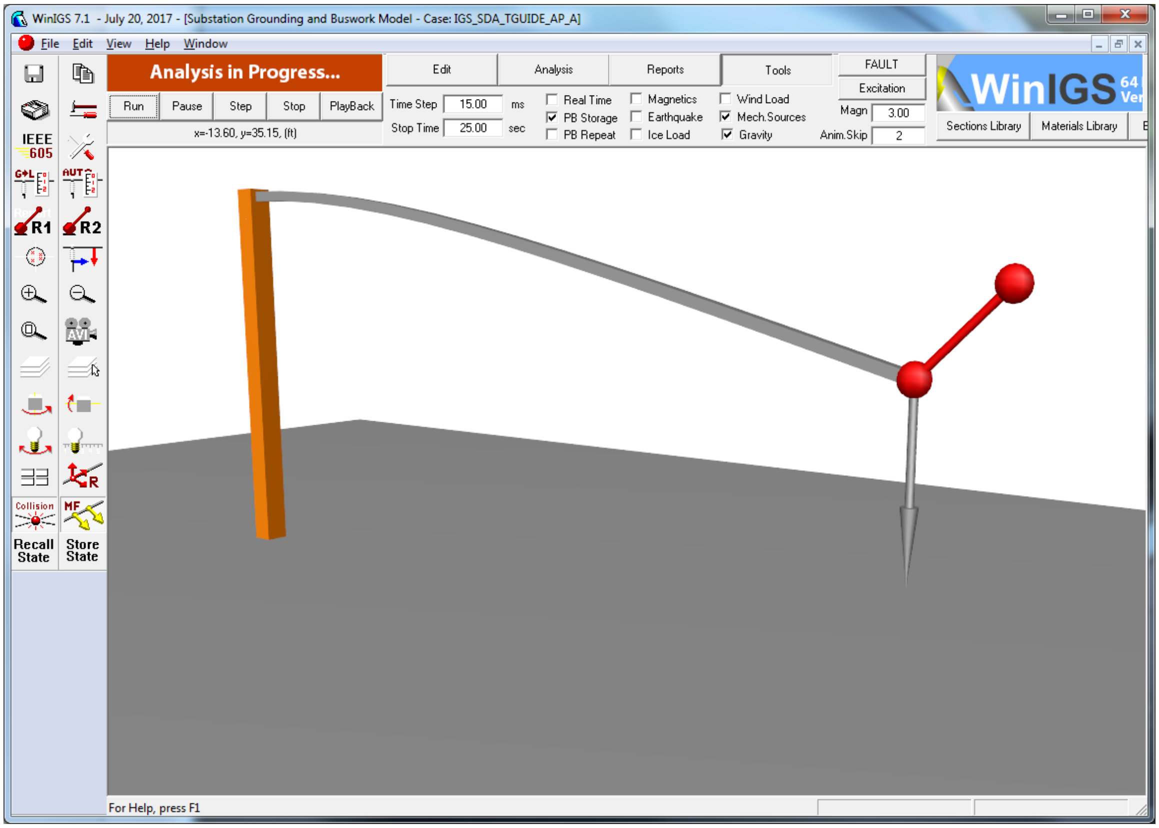


Figure A.16: Animated Rendered 3-D View Window

The 3-D rendered image view can be rotated and zoomed using the mouse. left click and drag the mouse over the window to rotate the point of view. User the mouse wheel to zoom in and out. The plot view display may be also panned and zoomed using the mouse left and right button or mouse wheel. Furthermore, one or two cursors may be activated that indicate numeric values of time and amplitude at the cursor locations. If two cursors are activated, the time difference between the cursors is also displayed. Use the buttons  to activate one or two cursors, respectively. Reposition one cursor at a time using the left mouse button, or move both cursors simultaneously using the right mouse button.

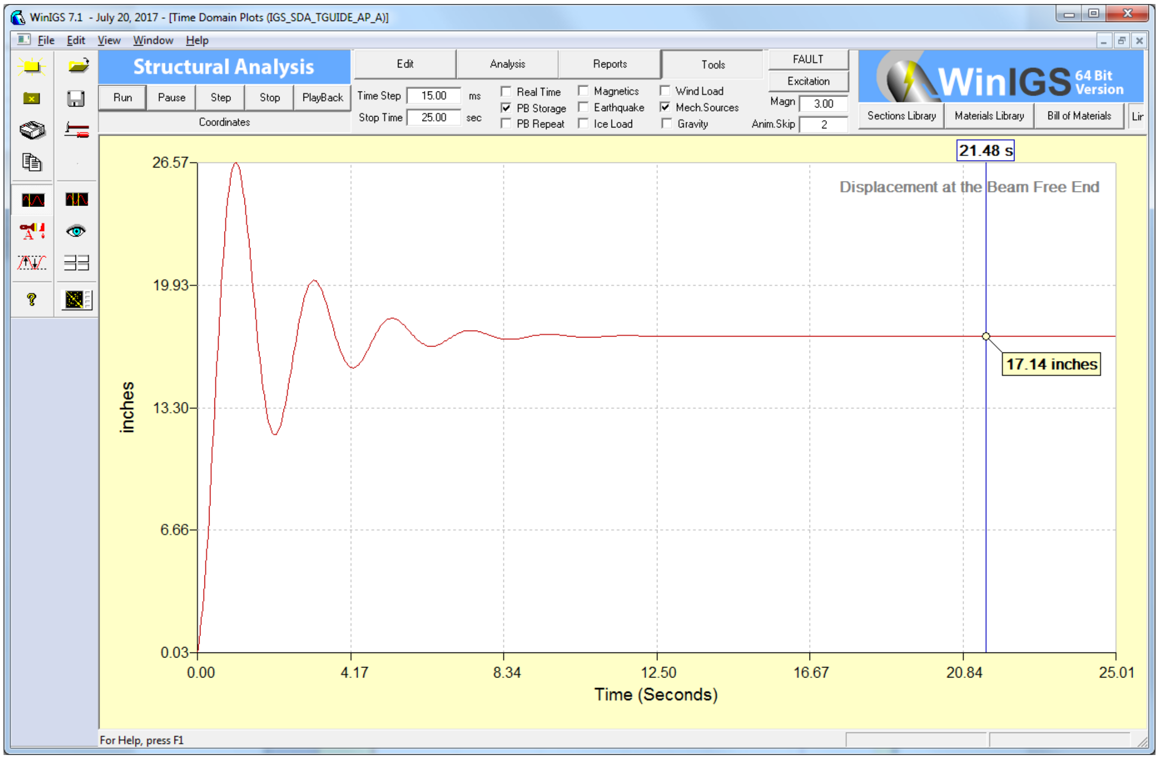
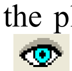


Figure A.17: Plot Window Showing Displacement of Beam Free End

The selection of the plotted quantities is made using the "**Select Plots**" form, which is opened using the  toolbar button, or the Select Meters command of the view menu. (Note that these commands are accessible only if a plot view window is the active window). The **Select Plots** form is illustrated in Figure A.18. The form contains two tables. The right table lists the available quantities to be plotted, while the left table lists the quantities that have been selected for plotting. Specific quantities can be added or removed from the selected quantities table (left), using the mouse to select the desired quantity and then clicking on one of the buttons: Add, Add to Group to add quantities and Remove or Remove Group to remove quantities. Note that the plot window may contain one or more plot frames, appearing one below the other. Quantities that are added within the same group are plotted in the same plot frame. Note also that this form contains two radio buttons that allow selection of ***English*** or ***Metric*** units.

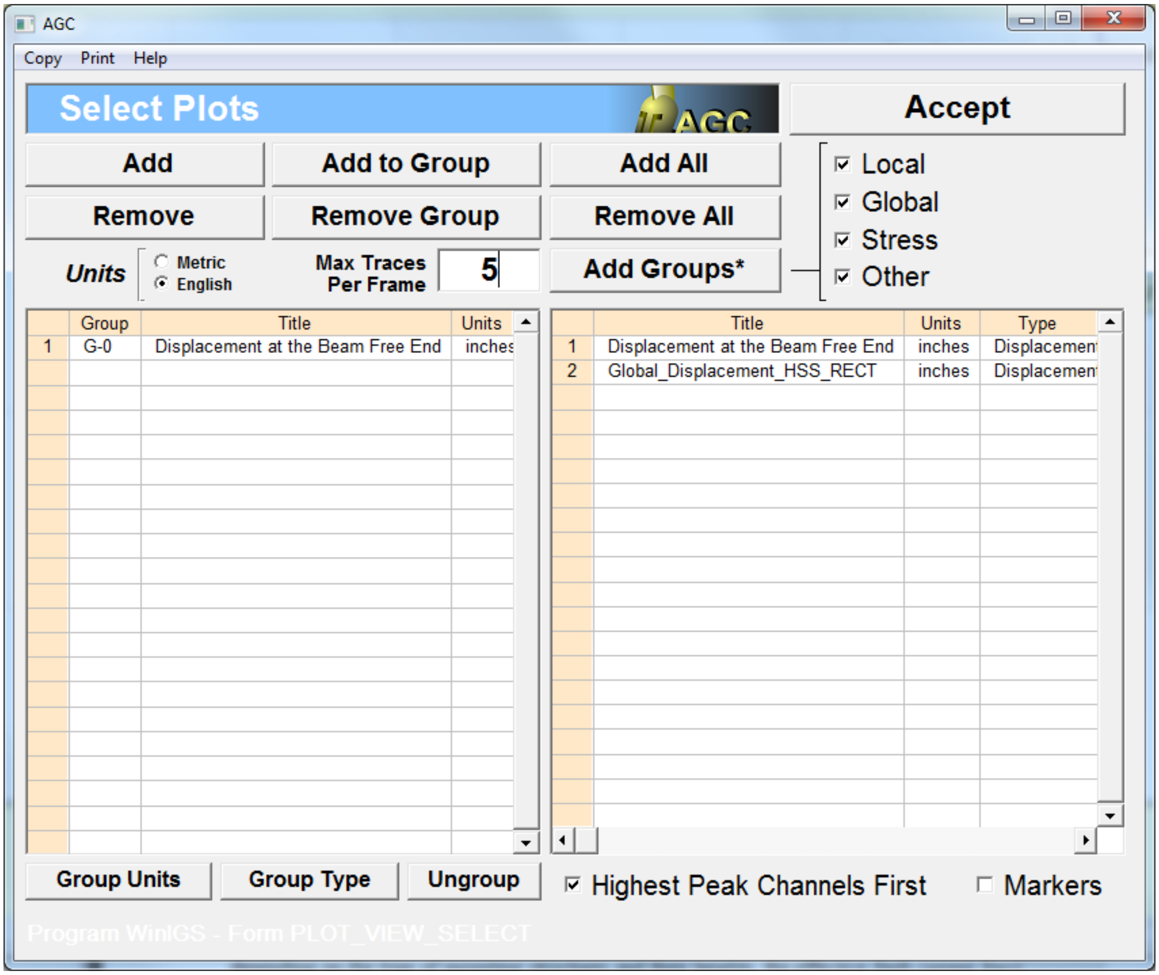


Figure A.18: Plot of Cantilever Beam End Displacement

A.4 Inspection of Results

When the simulation terminates, activate a single cursor, then drag the cursor near the right side of the plot frame to read off the numerical value of the beam free end displacement at steady state. The value at steady state for this example is **17.14 inches**, as illustrated in Figure A.17. Note that this value matches exactly the beam displacement computed manually in section A.3.

Next, execute a new analysis with the point source turned off and the Gravity excitation turned on. For this purpose, uncheck the **M. Sources** check box and check the **Gravity** check box. Note that the value at steady state is now **66.22 inches**, as illustrated in Figure A.19. This value closely matches the beam displacement computed via analytic methods (See section A.5), for the gravity excitation (uniformly distributed constant force equal to the beam weight).

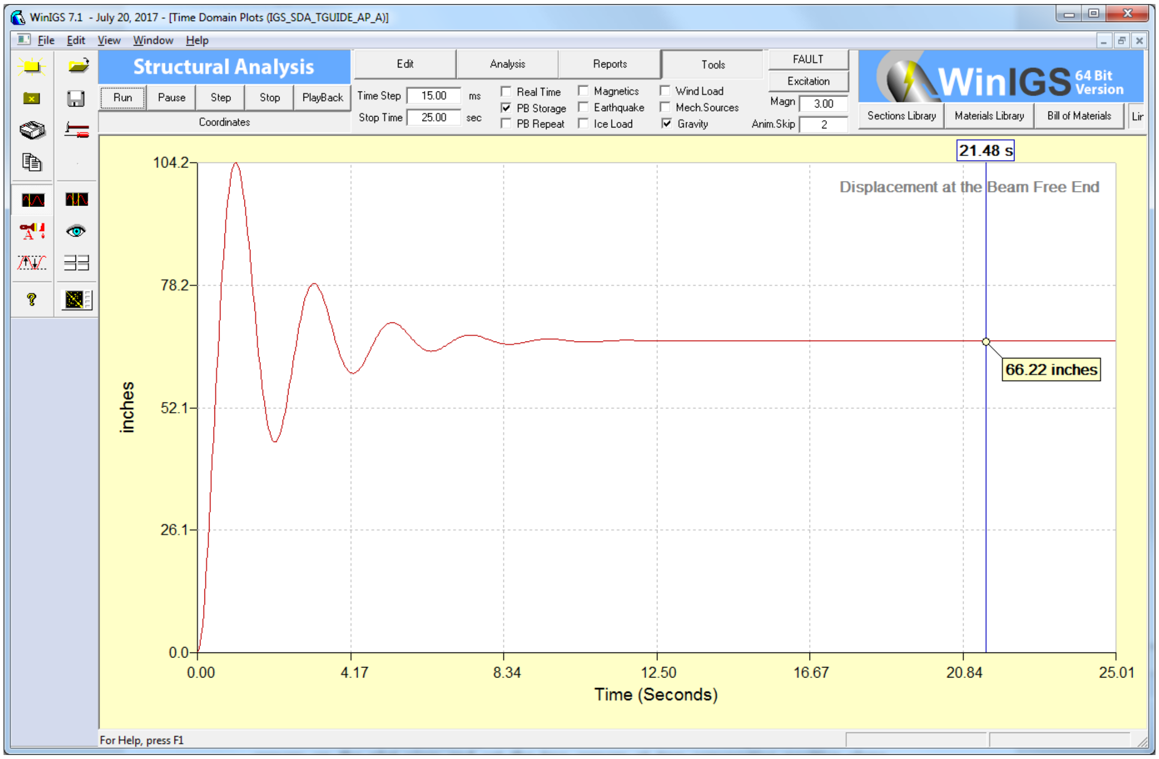


Figure A.19: Plot of Cantilever Beam End Displacement Under Gravity Excitation

To verify the first natural frequency of oscillation, execute a simulation with the damping ratio set to a low value, namely 0.05. When the simulation is completed, activate two cursors on the plot view and set the two cursors at two consecutive positive slope crossings of a value near the steady state value (approximately 17 inches), as illustrated in Figure A.20. Note that the indicated time difference between the two cursors approximates the oscillation period (2.091 seconds). Thus the frequency of oscillation is A.0 / 2.091 = **0.478 Hz**. This value closely matches the beam oscillation frequency computed using analytical methods (See Section A.5).

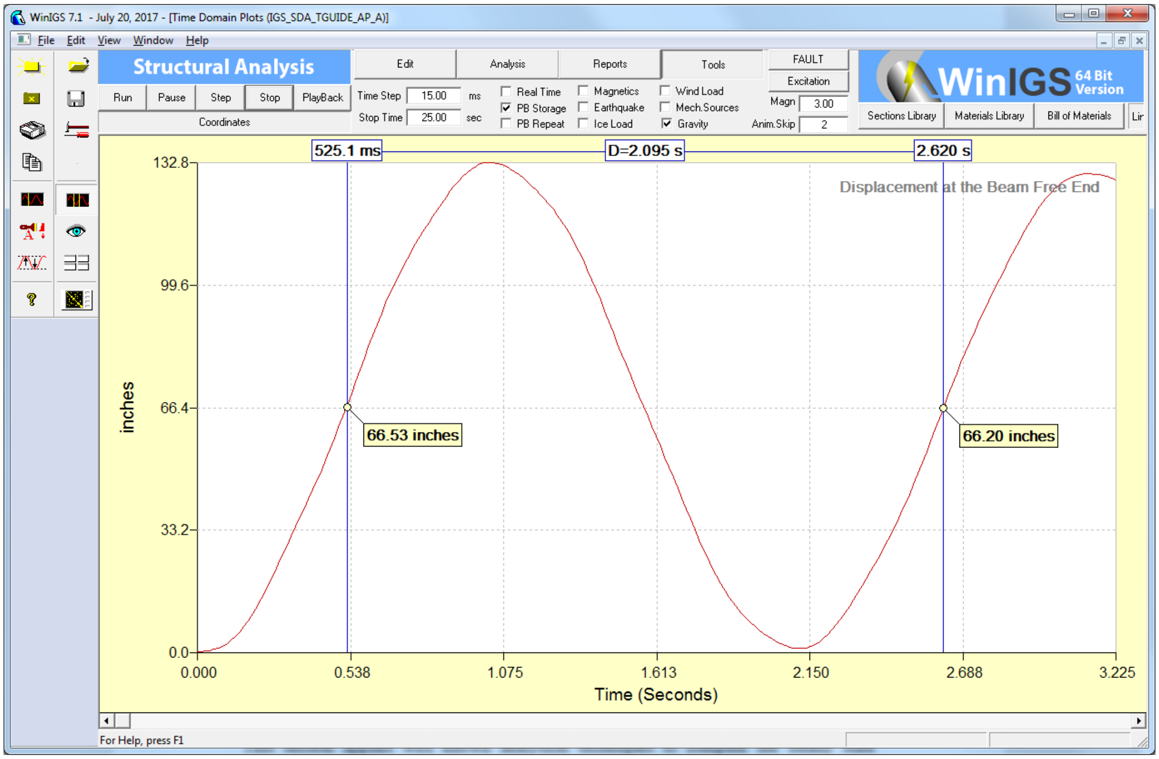


Figure A.20: Plot of Cantilever Beam End Displacement Period Measurement

A.5 Analytic Solution

This section applies well known analytical techniques to compute the steady state displacement and natural frequency of oscillation of the cantilever beam, for the purpose of validating the SDA solver. The displacement is computed at the cantilever beam free end, at steady state, i.e. after the oscillations have decayed. The natural frequency is computed assuming zero damping.

Displacement due to Concentrated Force

The displacement of the free end of the cantilever beam due to a concentrated force at the beam free end can be computed analytically using the equation:

$$x = \frac{F\ell^3}{3EI}$$

where:

F is the applied force magnitude (1000lb),
ℓ is the beam length (100 feet),

E is the material of the modulus of elasticity (for steel E = 2.05×10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (1130 inches^4)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$x = \frac{1000lb \times 4.4482\frac{N}{lb} \times \left(100ft \times 0.3048\frac{m}{ft}\right)^3}{3 \times 2.05 \times 10^{11}\frac{N}{m^2} \times 1130\, inch^4 \times \left(0.0254\frac{m}{inch}\right)^4} = 0.43545m = 17.14"$$

Thus the application of a 1000 lb force at the end of the beam results in a 17.14 inch displacement at the beam free end. *Note that gravity is ignored in this example*.

Displacement due to Gravity

The displacement of the free end of the cantilever beam due to gravity can be computed analytically using the equation:

$$x = \frac{mg\ell^3}{8EI} = \frac{w\ell^4}{8EI}$$

where:

m is the beam mass
w is the weight of the beam per unit of length (in N/m)
g is the acceleration of gravity (9.80665 m/sec^2)
$\boldsymbol{\ell}$ is the beam length (in meters),
E is the material of the modulus of elasticity (for steel E = 2.05×10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (1130 inches^4)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$x = \frac{1503.2\, N/m \times \left(100ft \times 0.3048\frac{m}{ft}\right)^3}{8 \times 2.05 \times 10^{11}\frac{N}{m^2} \times 1130\, inch^4 \times \left(0.0254\frac{m}{inch}\right)^4} = 1.6820m = 66.22"$$

Thus under its own weight the beam free end displacement will be 66.22 inches.

First Natural Frequency

The first natural frequency of the beam is computed analytically using the equation:

$$f = \frac{1.875^2}{2\pi}\sqrt{\frac{E \times I}{m \times \ell^3}} = \frac{1.875^2}{2\pi}\sqrt{\frac{g \times E \times I}{w \times \ell^4}}$$

where:

m is the mass of the beam (7850kg)
g is the acceleration of gravity (9.80665 m/sec^2)
w is the weight of the beam per unit of length (N/m)
$\boldsymbol{\ell}$ is the beam length (in meters),
E is the material of the modulus of elasticity (for steel E = 2.05x10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (4.70342 * 10^{-4} meters^4)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$f = \frac{1.875^2}{2\pi}\sqrt{\frac{9.80665 \times 2.05 \times 10^{11} \times 4.70342 \times 10^{-4}}{1503.2 \times 30.48^4}} = 0.47767\ Hz$$

Applying a constant force at the end of the beam will mainly excite the first natural frequency thus the period of oscillation will be approximately equal to 1.0 / 0.477 Hz = 2.093 seconds.

The solution of this example problem using the WinIGS SDA solver is presented next. The beam free end displacement and the frequency of oscillation computed in this section are compared with the corresponding WinIGS results.

A.6 Discussion

This example provided a validation procedure of the WinIGS SDA solver by comparing the steady state response of a cantilever elastic beam computed by WinIGS to the value computed using well known equations. Note however that the Dynamic solver provides a much more comprehensive description of the system behavior that a static solution. In fact the analysis shows that the maximum beam deflection is substantially higher (about 24.43 inches) occurring about one second after the application of the force. Furthermore the dynamic analysis also provides several other quantities of interest such as stresses at any point of the beam as functions of time. Comparing the maximum stress values to corresponding material limits, the adequacy of the system design can be evaluated.

Appendix B. Simply Supported Beam

B.1 Introduction

This section presents the dynamic analysis of a single steel beam simply supported at the two ends. The main objective of this example is to provide a validation benchmark of the WinIGS structural dynamics solver. The WinIGS data files for the example system are provided under the study case name:

IGS_SDA_AGUIDE_CH02

The steel beam characteristics and support configuration are illustrated in Figure B.1. The beam has hollow square cross-section. Both ends of the beam are simply supported (i.e. the end point positions are fixed but their rotations are free). The displacement at the center point of the beam is computed using analytic formulas and the results are compared to the WinIGS simulation results. The displacement is computed for two cases: (a) A 1000 lb vertical concentrated force is applied at the center of the beam, while the weight of the beam is ignored, and (b) The weight of the beam is taken into account (no concentrated force applied). Furthermore, the first natural frequency of the beam is computed and compared to the frequency of oscillation during the dynamic simulation.

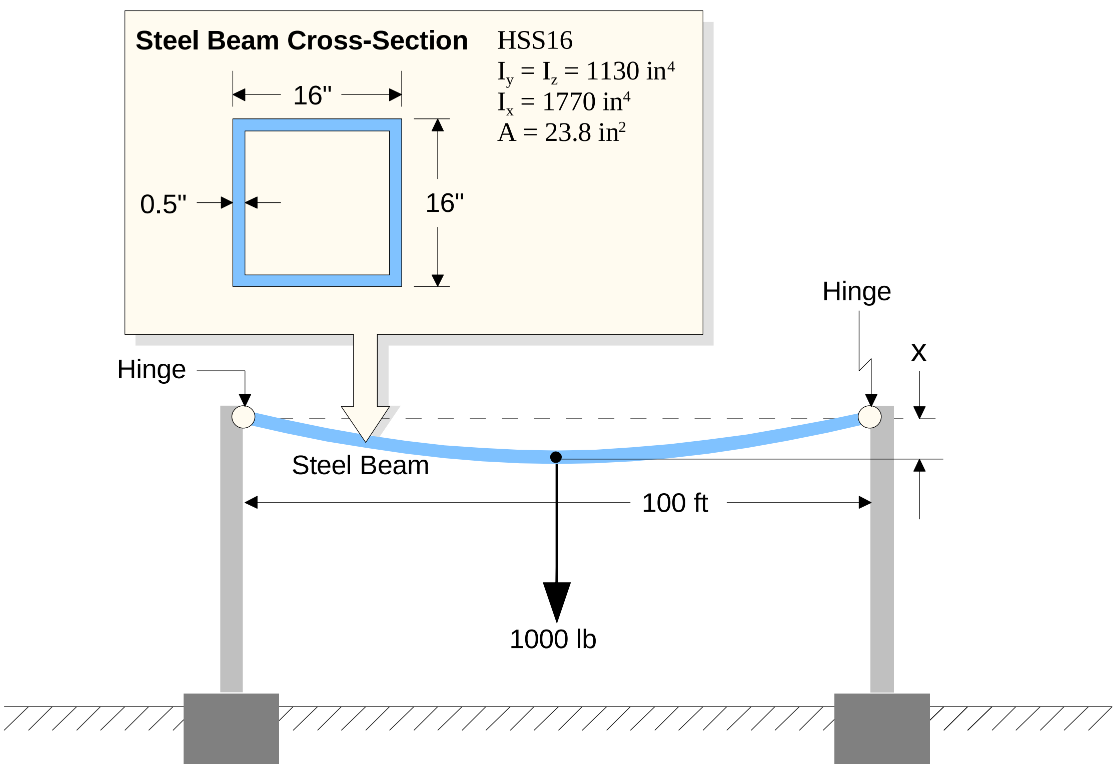
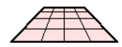


Figure B.1: System Configuration

B.2 Inspection of System Data

In order to run this example, execute the program WinIGS and open the study case titled: IGS_SDA_TGUIDE_AP_B. Use command **Open** of the **File** menu or click on the icon:  to open the existing study case data files. Note that the example study case data files are placed in the directory \IGS\DATAU during the WinIGS program installation.

Once the study case files are opened, the network editor window appears showing the system single line diagram, illustrated in Figure B.1. As in chapter 1, this example case does not include an electrical network, and therefore the only model element defined is a "***Grounding System Element***" represented by the icon:  which contains the simply supported beam model. Double click on the ***Grounding System Element*** icon to open the grounding system editor and examine the beam model. The top view, side view, and 3-D rendered view of the modeled system is illustrated in Figures B.3, B.4, and B.5.

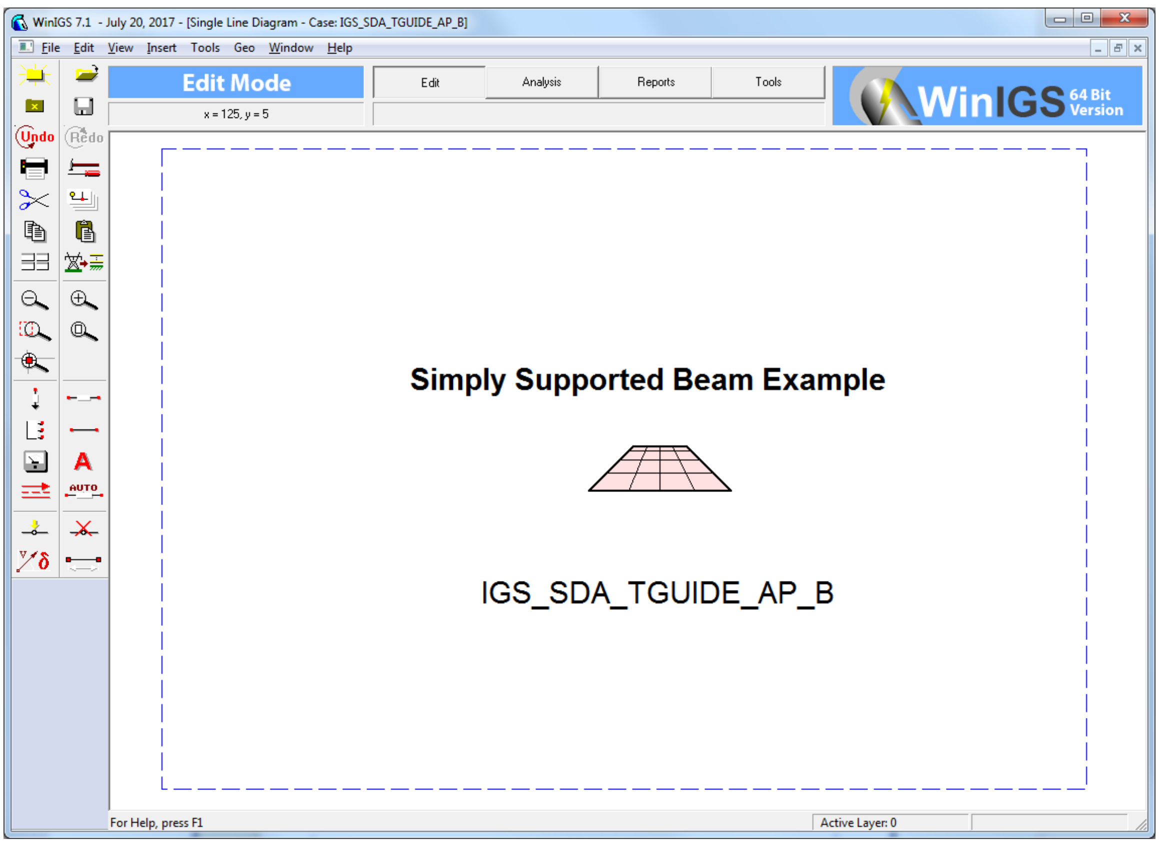


Figure B.2 Single Line Diagram of Example System IGS_SDA_AGUIDE_CH02

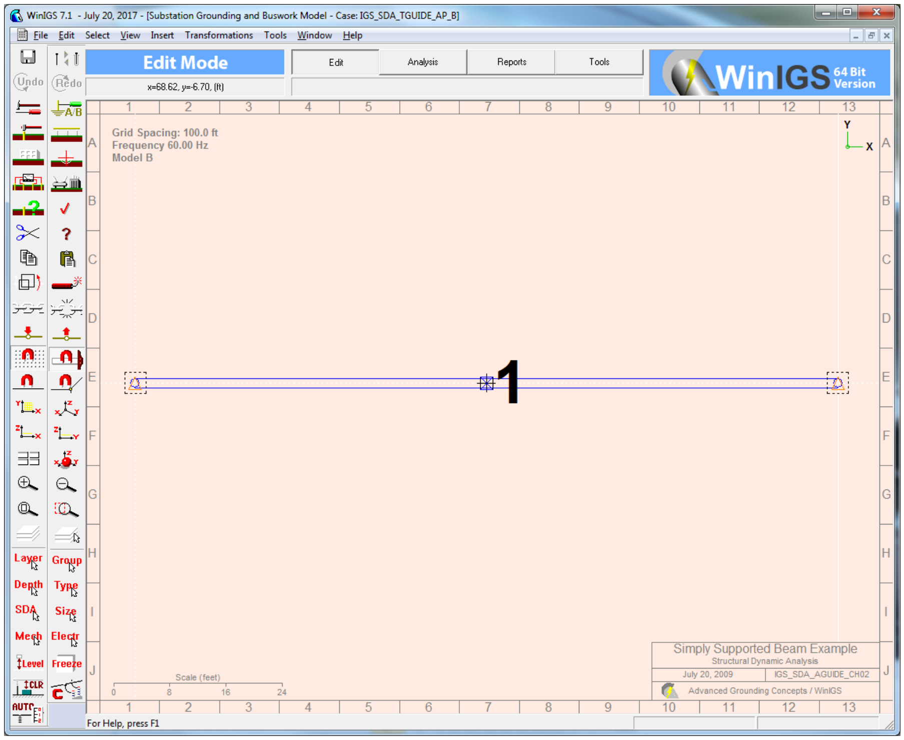


Figure B.3: Simply Supported Beam Model – Top View

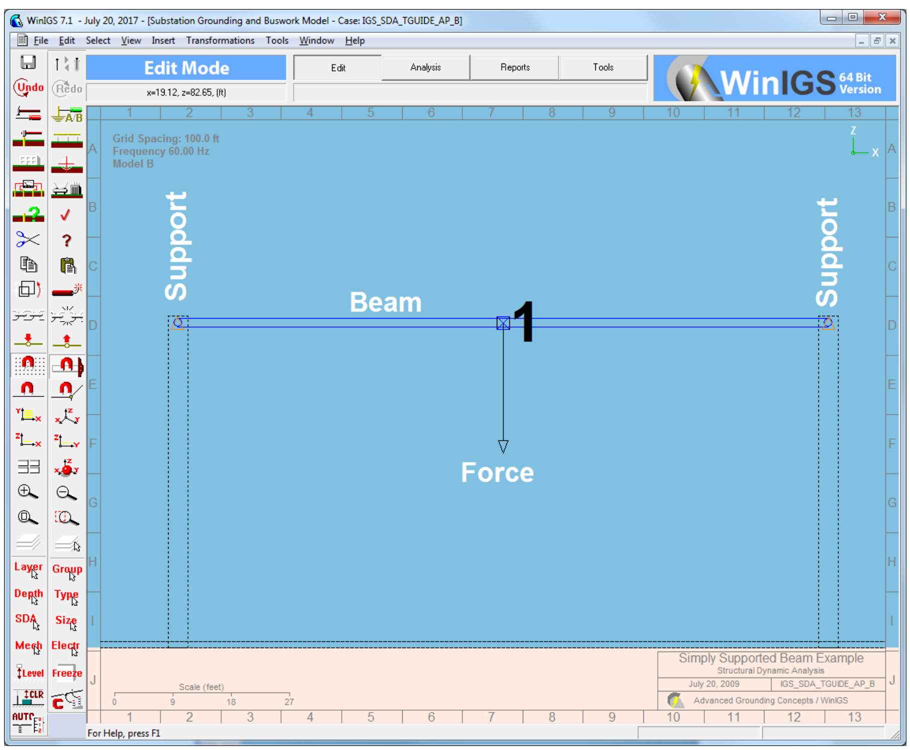


Figure B.4: Simply Supported Beam Model – Side View

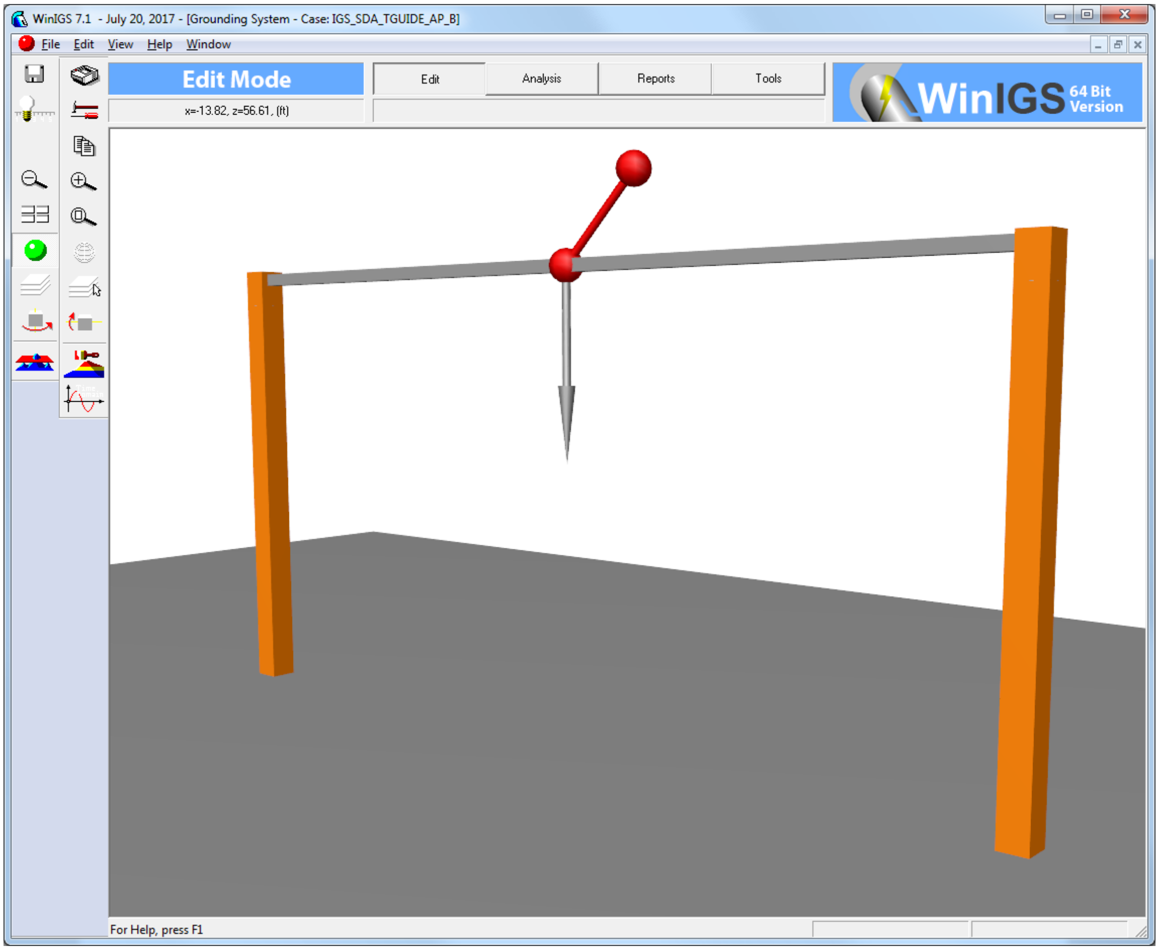


Figure B.5: Simply Supported Beam Model – Rendered Perspective View

Next, examine the parameters of each model element. Double click on each element to open the corresponding parameter form. For detailed descriptions of the parameters of the model elements of this example (beam, two support elements, force, and displacement meter) please refer to chapter 1. The parameter forms of the main elements comprising this model are illustrated in Figures B.6, B.7, B.8, B.9, B.10, and B.11.

Note that the support elements used in this example (See Figures B.9 a and b) have the rotation support condition check boxes deactivated. This simulates the operation of hinged supports rather than fixed supports.

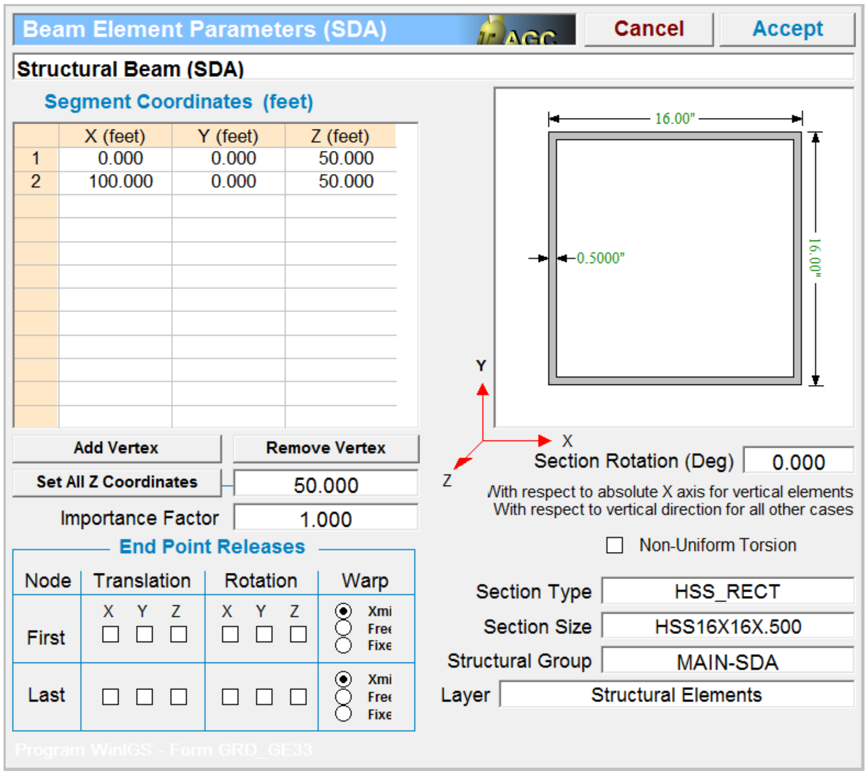


Figure B.6: Beam Parameters

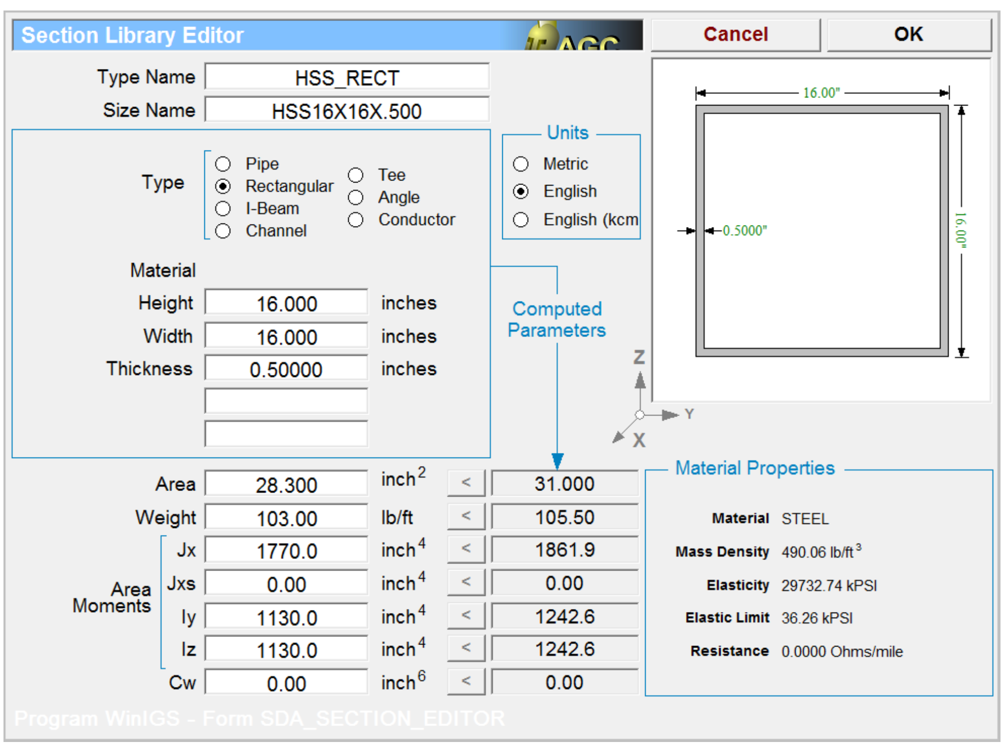
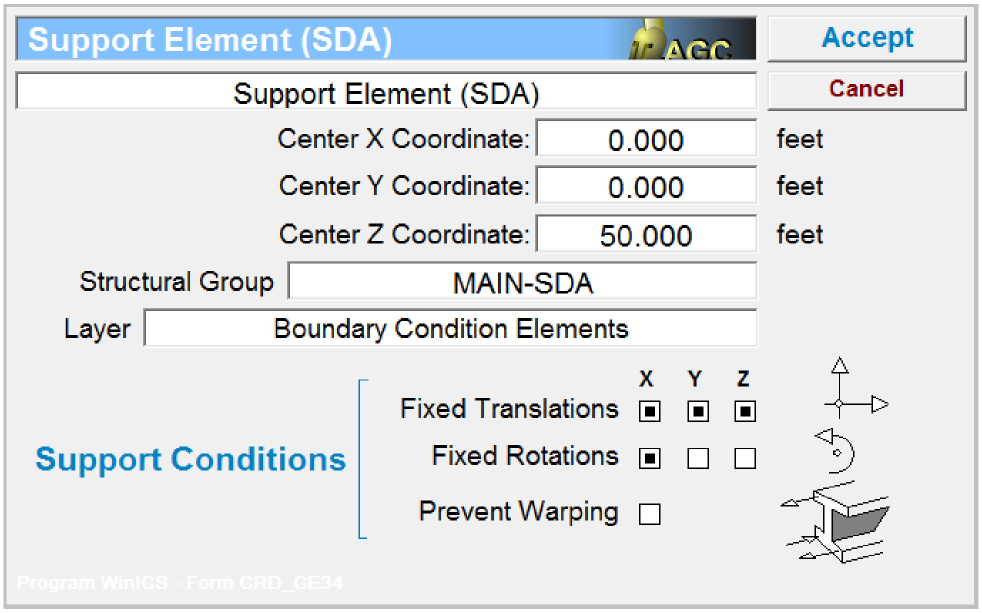
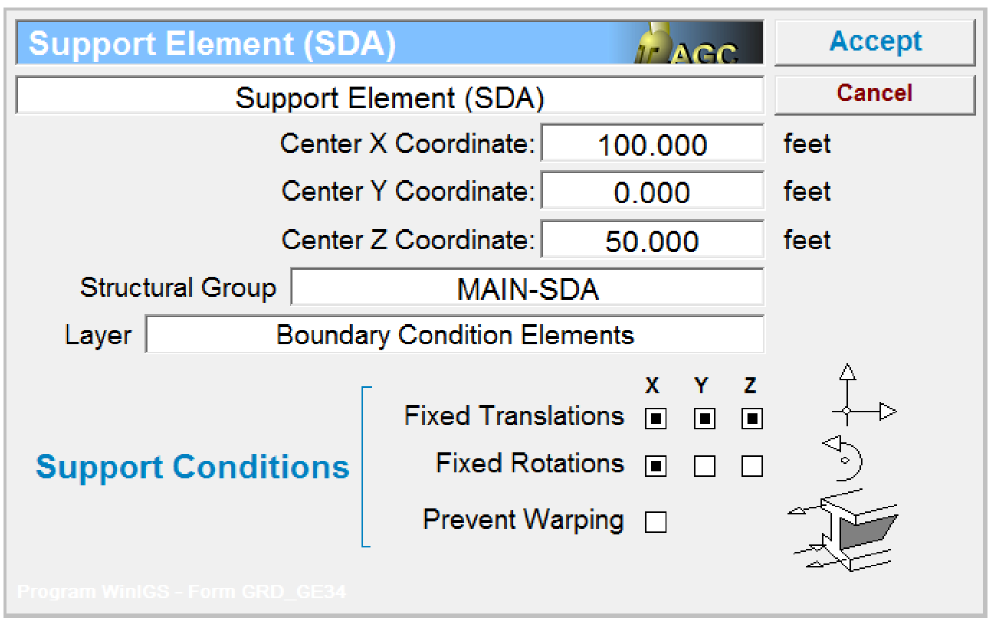


Figure B.8: Section Parameters



(a)



(b)

Figure B.9: Support Element Parameters

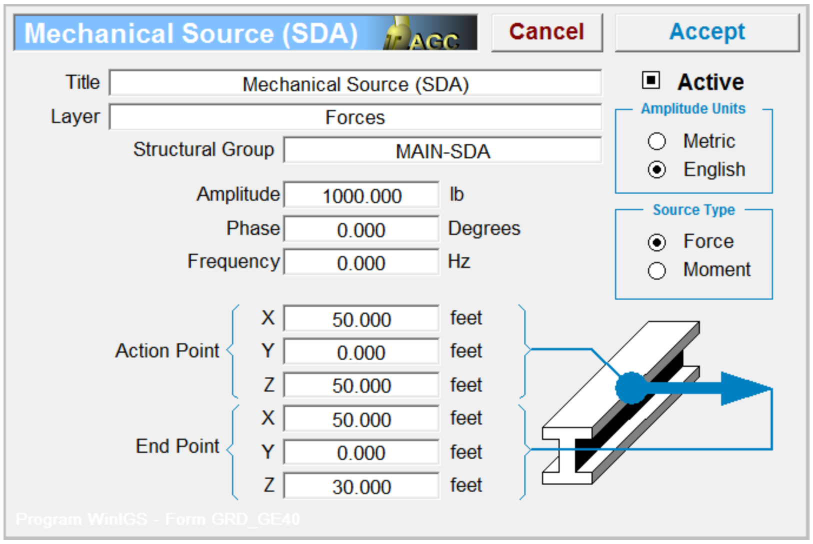


Figure B.10: Mechanical Source Parameters

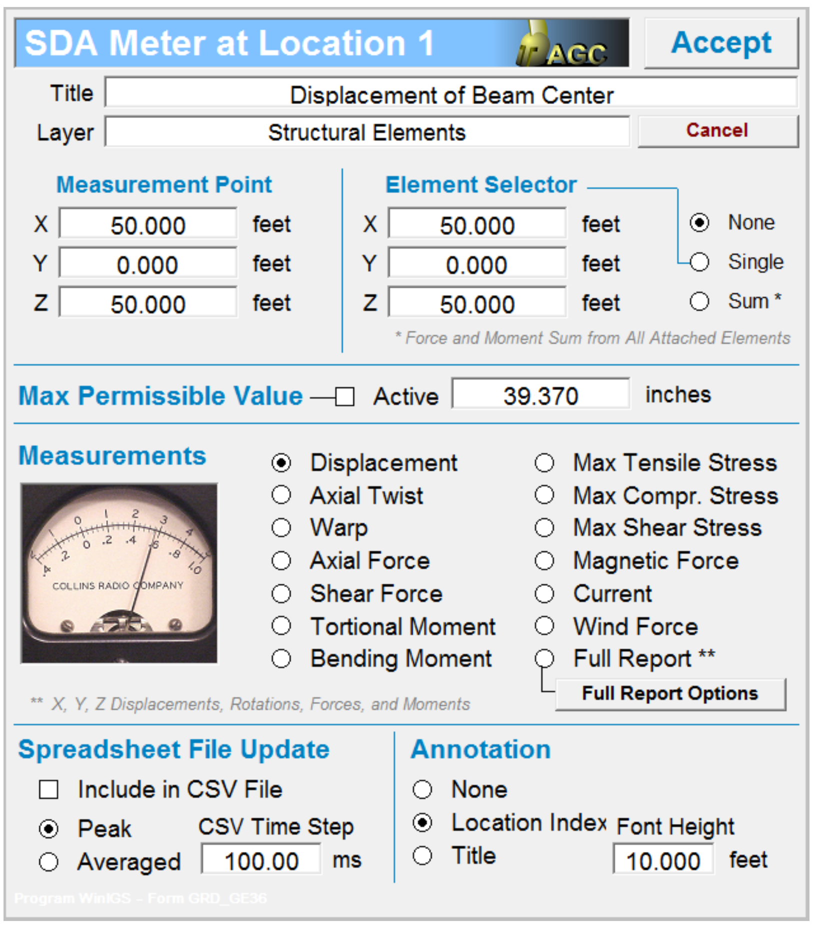


Figure B.11: Structural Dynamics Meter Parameters

B.3 Analytic Solution

This section applies well known analytical techniques to compute the steady state displacement and natural frequency of oscillation of the beam element for the purpose of validating the SDA solver. The displacement is computed at the beam center point at steady state, i.e. after the oscillations have decayed. The natural frequency is computed assuming zero damping.

B.3.1 Displacement due to Concentrated Force

The displacement of the center point of a simply supported beam due to a concentrated force at the beam center can be computed analytically using the equation:

$$x = \frac{F\ell^3}{48EI}$$

where:

F is the applied force magnitude (1000lb),
$\boldsymbol{\ell}$ is the beam length (100 feet),
E is the material of the modulus of elasticity (for steel E = 2.05×10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (1130 $inches^4$)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$x = \frac{1000lb \times 4.4482\frac{N}{lb} \times \left(100ft \times 0.3048\frac{m}{ft}\right)^3}{48 \times 2.05 \times 10^{11}\frac{N}{m^2} \times 1130\, inch^4 \times \left(0.0254\frac{m}{inch}\right)^4} = 0.02721\, m = 1.071''$$

Thus the application of a 1000 lb force at the beam center results in a **1.071** inch displacement at the beam center. *Note that gravity is ignored in this example*.

B.3.2 Displacement due to Gravity

The displacement of the center point of a simply supported beam due to gravity can be computed analytically using the equation:

$$x = \frac{5mg\ell^3}{384EI}$$

where:

m is the beam mass (103 lb/ft x 100 ft / (2.20461 lb/kg) = 4672kg)
g is the acceleration of gravity (9.80665 m/sec^2)
ℓ is the beam length (100 feet),
E is the material of the modulus of elasticity (for steel E = 2.05x10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (1130 inches^4)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$x = \frac{5 \times 4672\,kg \times 9.80665\,\frac{m}{sec^2} \times \left(100ft \times 0.3048\,\frac{m}{ft}\right)^3}{384 \times 2.05 \times 10^{11}\,\frac{N}{m^2} \times 1130\,inch^4 \times \left(0.0254\,\frac{m}{inch}\right)^4} = 0.17520\,m = 6.898''$$

Thus under its own weight the beam center point displacement will be **6.898** inches.

B.3.3 Natural Frequency

The first natural frequency of a simply supported beam is computed analytically using the equation:

$$f = \frac{\pi}{2}\sqrt{\frac{E \times I}{m \times \ell^3}}$$

where:

m is the mass of the beam (7850kg/m^3 x 100ft x 0.3048 x 28.3inch^2 x 0.0245^2 = 4368kg)
ℓ is the beam length (100 feet),
E is the material of the modulus of elasticity (for steel E = 2.05x10^{11} N/m^2), and
I is the beam section moment of inertia about the axis perpendicular to the beam main axis (1130 inches^4)

Applying the numerical values related to this example yields (all quantities are converted to MKSA units):

$$f = \frac{\pi}{2}\sqrt{\frac{2.05 \times 10^{11}\,\frac{N}{m^2} \times 1130\,inch^4 \times \left(0.0254\,\frac{m}{inch}\right)^4}{4672\,kg \times \left(100ft \times 0.3048\,\frac{m}{ft}\right)^3}} = 1.341\,Hz$$

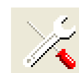
Applying a constant force at the beam center will mostly excite the first natural frequency thus the period of oscillation will be approximately equal to 1.0 / 1.341 Hz = **0.746** seconds.

The solution of this example problem using the WinIGS SDA solver is presented next. The beam displacement and the frequency of oscillation computed in this section are compared with the corresponding WinIGS results.

B.4 Numerical Solution using WinIGS

In order to access the Structural Dynamic Analysis solver, click on the Tools Button (top row of WinIGS buttons), click on the grounding system symbol that contains the structural model, and then click on the **Mechanical** button.

B.4.1 Mechanical Analysis Parameters

Before executing the time domain simulation, click on the toolbar button  to open the simulation parameters window, illustrated in Figure B.12. (Also accessible via the Mechanical Analysis Parameters command of the Tools pull-down menu).

Ensure that the selected options are as indicated in Figure B.12. For a detailed description of the mechanical analysis parameters, please refer to Appendix A. Also ensure that the main WinIGS toolbar controls are set as illustrated in Figure B.13.

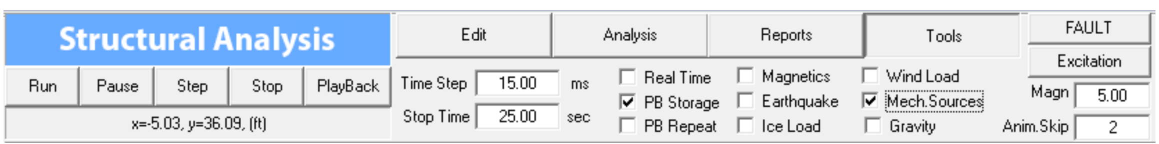


Figure B.13: Main SDA Main Toolbar Settings

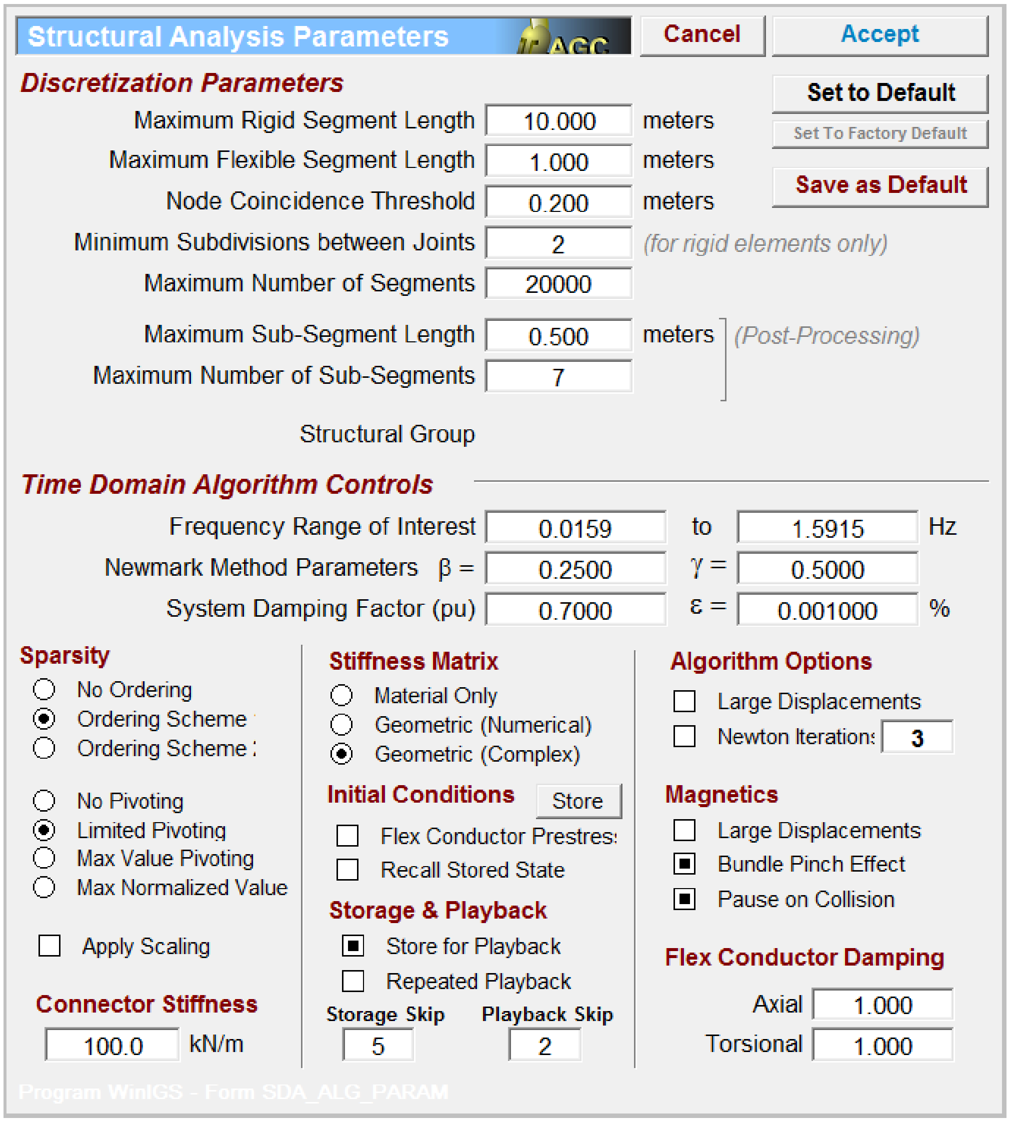


Figure 1.12: Mechanical Analysis Parameters

B.4.2 Running the Dynamic Analysis

After inspection of the system parameters click on the START button of the main SDA toolbar to execute the dynamic analysis. Upon initiation of the analysis, a 3-D rendered window (shown in Figure B.14) and a plot window are automatically opened.

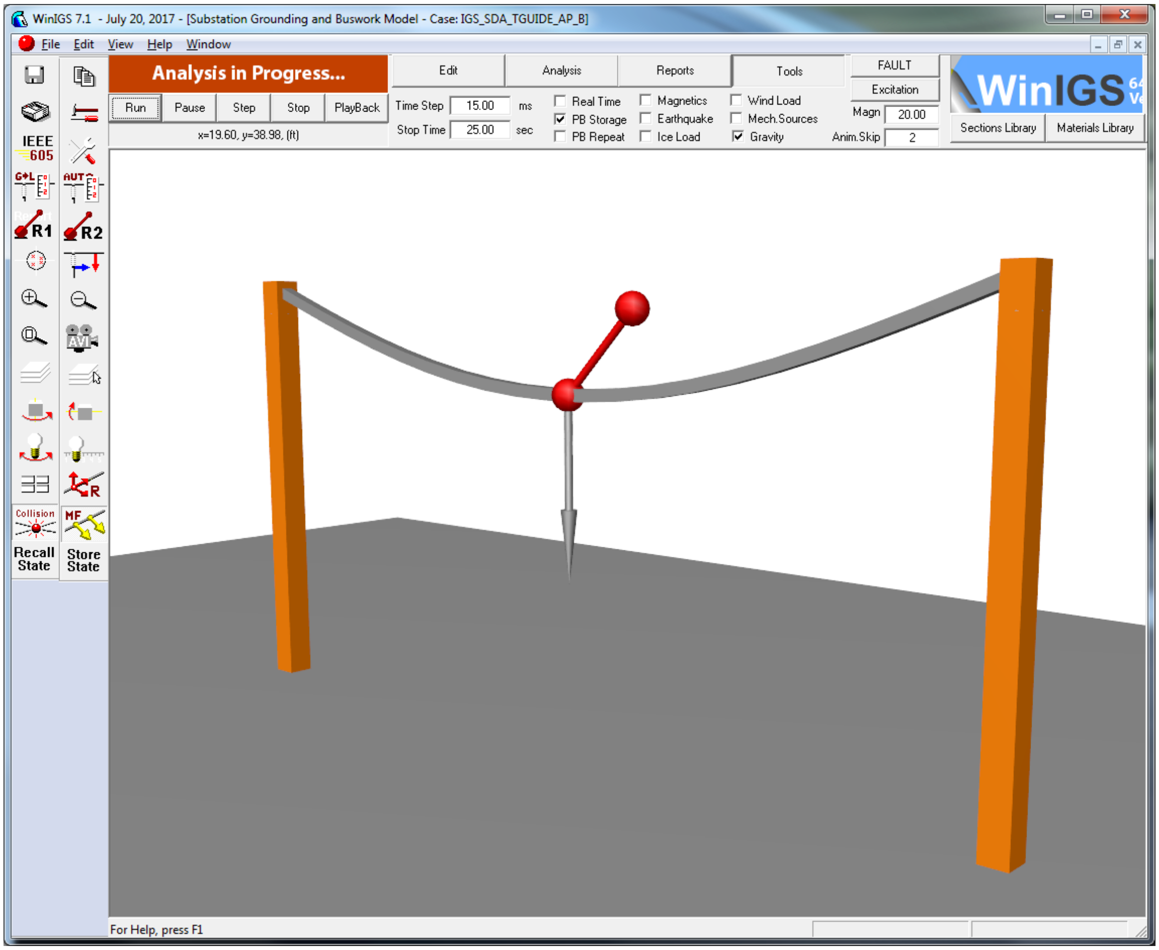


Figure B.14: Animated Rendered 3-D View Window

B.5 Inspection of Results

When the simulation terminates, activate a single cursor, then drag the cursor near the right side of the plot frame to read off the numerical value of the beam center point displacement at steady state. The value at steady state for this example is **1.071 inches**, as illustrated in Figure 1.15. Note that this value matches exactly the beam displacement computed manually in section B.3. (For a detailed description on using the plot view user interface controls, please refer to chapter 1, section 1.5

Next, execute a new analysis with the point source turned off and the Gravity excitation turned on. For this purpose, uncheck the **M. Sources** check box and check the **Gravity** check box. Note that the value at steady state is now **6.45 inches**, as illustrated in Figure B.16. Note that this value matches exactly the beam displacement computed manually in section B.3, due to the influence of gravity.

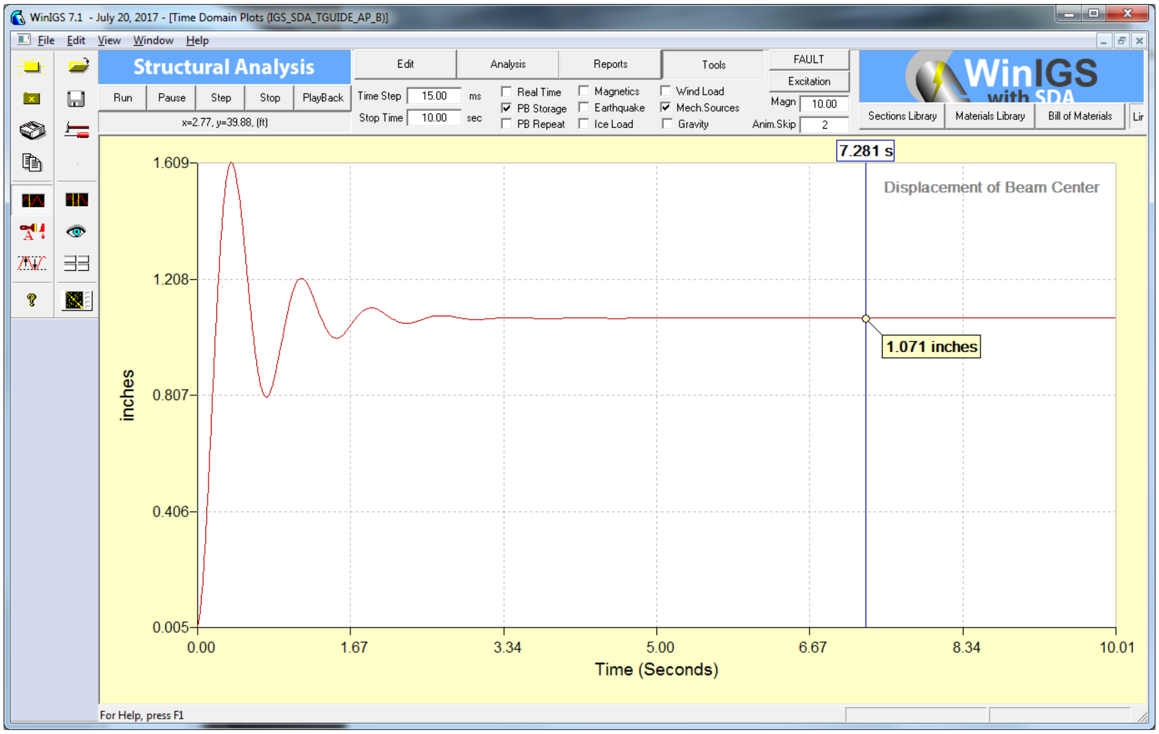


Figure B.15: Plot Window Showing Displacement of Beam Center Point

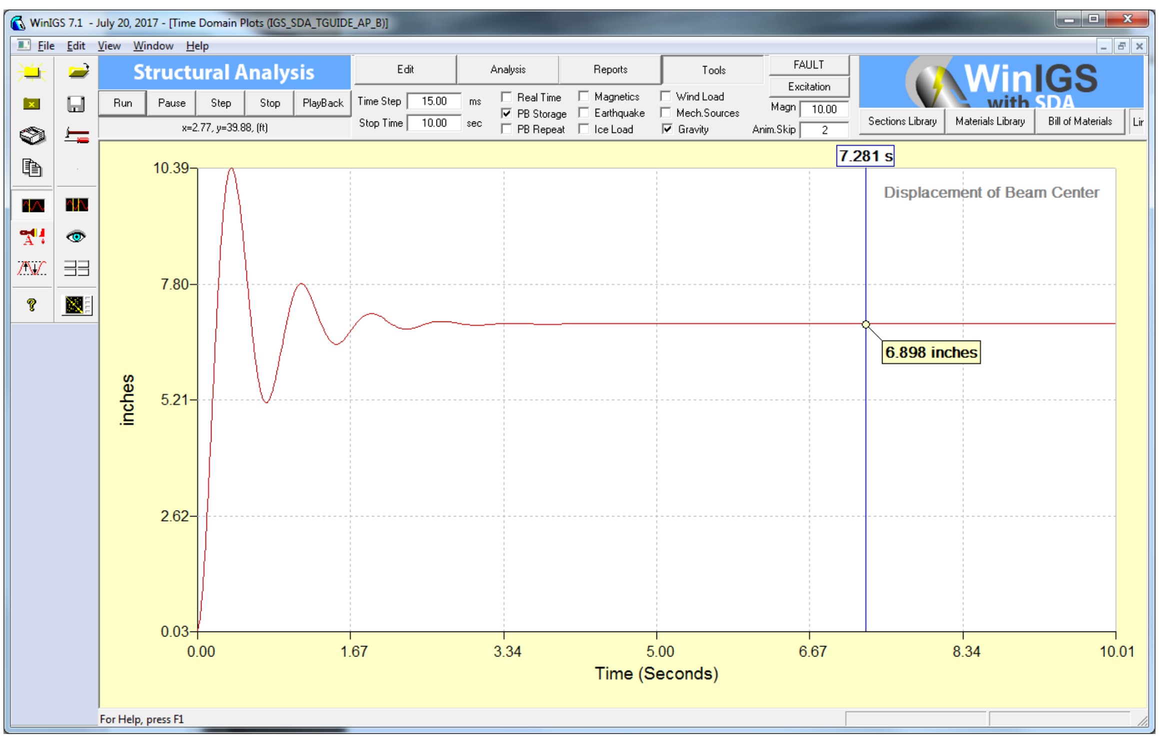


Figure B.16: Plot of Beam Center Point Displacement Under Gravity Excitation

To verify the first natural frequency of oscillation, execute a simulation with the damping ratio set to a low value, namely 0.05. When the simulation is completed, activate two cursors on the plot view and set the two cursors at two consecutive positive slope crossings of a value near the steady state value (approximately 1.04 inches), as illustrated in Figure B.17. Note that the indicated time difference between the two cursors approximates the oscillation period (**0.752 seconds**). Thus the frequency of oscillation is 1.0 / 0.752 = **1.33 Hz**. Note that this value closely matches the beam oscillation frequency computed manually in section B.3.
An alternative tool for computing resonance frequencies is the Eigenvalue Analysis tool. The eigenvalue analysis results for this example are illustrated in Figure B.18. Note that the first (lowest) computed resonance frequency exactly matches the analytical result (1.341 Hz).

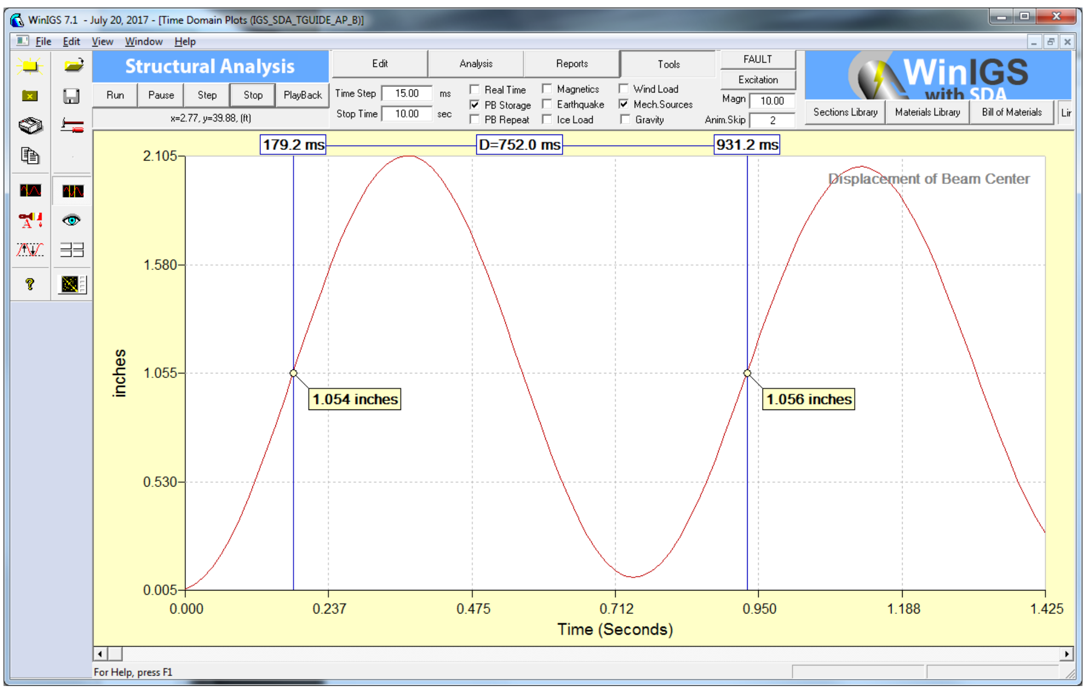


Figure B.17: Measurement of Beam Center Point Displacement Period

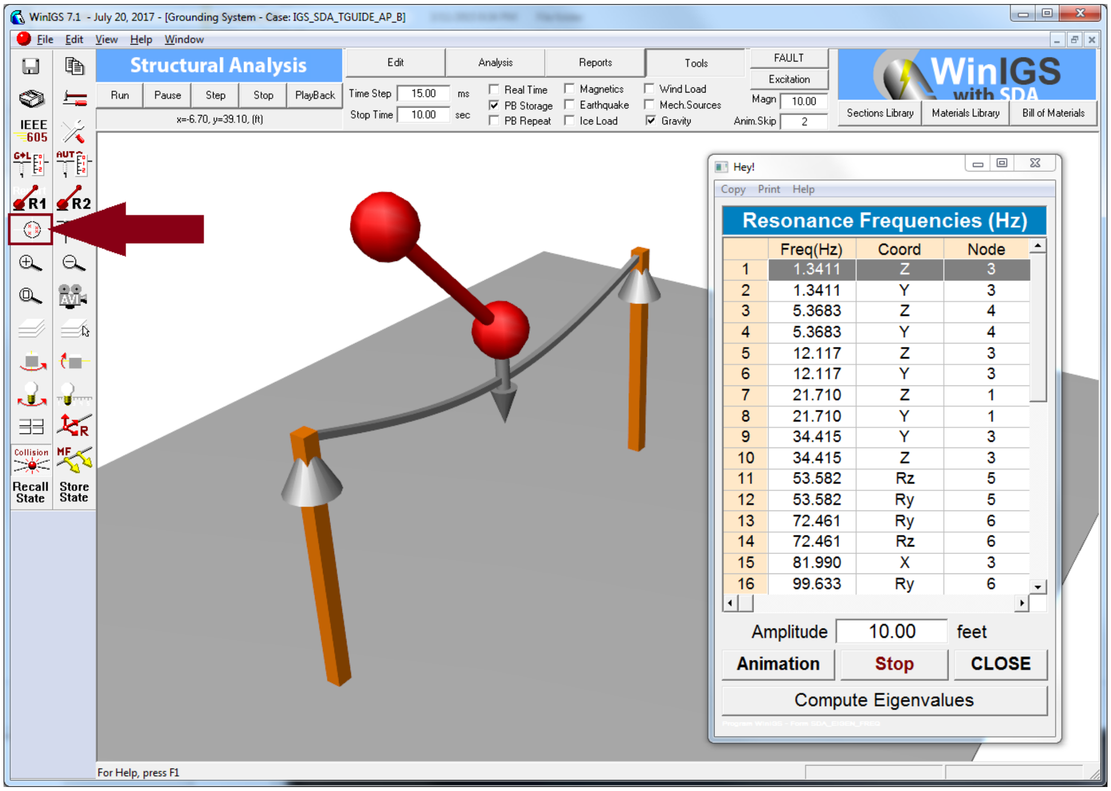


Figure B.18: Direct Computation of Resonance Frequencies Using the Eigenvalue Command (Indicated Toolbar Button)

B.6 Discussion

This example provided a validation procedure of the WinIGS SDA solver by comparing the steady state response of a simply supported elastic beam computed by WinIGS to the value computed using well known analytic techniques. Note however that the Dynamic solver provides a much more comprehensive description of the system behavior that a static solution. In fact the analysis shows that the maximum beam deflection is substantially higher, namely about 1.7 inches under the concentrated force excitation and 10.5 inches due to the beam weight. These maximum values occur about 0.35 seconds after the application of the force. Furthermore the dynamic analysis also provides several other quantities of interest such as stresses at any point of the beam as functions of time. Comparing the maximum stress values to corresponding material limits, the adequacy of the system design can be evaluated.

Appendix C. Insulator Modeling

Given manufacturer insulator specs, there are two ways to setup section and material parameters for insulator modeling. If the cantilever strength is specified, then the max allowable **Bending Moment** computed from the **Max Cantilever Force** can be entered as a **section** property. From this entry WinIGS automatically computes the max permitted **Stresses**. Alternatively, if the maximum permitted **Tensile** force is specified by the manufacturer, then a material can be added using the material library editor with the **Elastic Limit** property set equal to the permissible tensile stress (computed as tensile force over section area). If the maximum permissible **Torsion** and **Compression Force** are also specified, the shear and compression to tensile stress ratios can be computed and entered in the section properties of the insulator model.

The above described procedures are illustrated by example in the next two sections, using the example manufacturer specifications shown in Figure C.1.

Diameter	6.25 inches
Height	24 inches
Cantilever Strength	4000 lb
Tensile Strength	25,000 lb
Compression Strength	60,000 lb
Torsional Strength	20,000 inch-lb

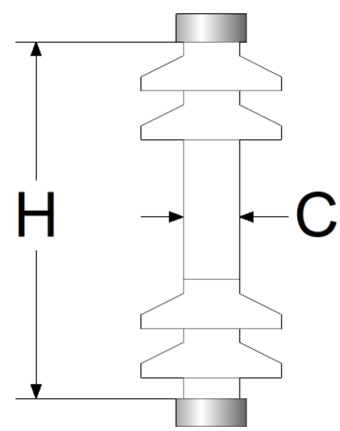


Figure C.1: Example Manufacturer Specifications

C.1: Using the Cantilever Strength Specification.

In the example specs shown in Figure C-1, the cantilever strength is specified at 4000 lb. for a 24 inch long insulator. Assuming that insulator is vertically oriented, rigidly supported at the bottom, and a 4000 lb. horizontal force is applied at the top, then the maximum bending moment will occur at the bottom end of the insulator. The maximum bending moment will be equal to the applied force multiplied by the insulator length, i.e.: 8000 lb-ft (see Figure C.2)

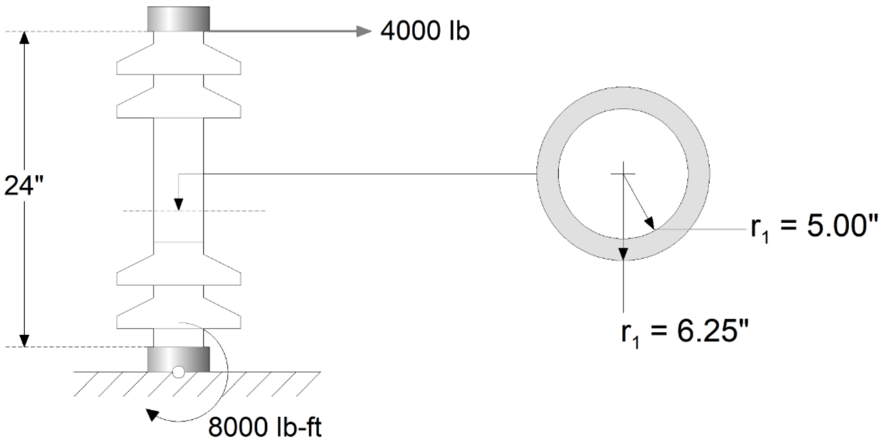


Figure C.2: Example insulator with bending moment due to applied cantilever force.

Using the WinIGS section editor, a section is added to the library as shown in Figure C.3. The insulator porcelain cylinder inner and outer diameters (5.00” and 6.25”) are entered in the corresponding fields. Using the cylinder dimensions, WinIGS automatically computes the geometric section parameters, namely: the cross-sectional area, weight, and moments of inertia. These results are listed under the “**Computed Parameters**” title. *Click on the arrow buttons (<) next to each computed geometric value to copy the computed values into the corresponding entry fields located to the left of the computed values.* Alternatively, if there are known geometric parameters you can type them directly into these entry fields, overriding the computed values.

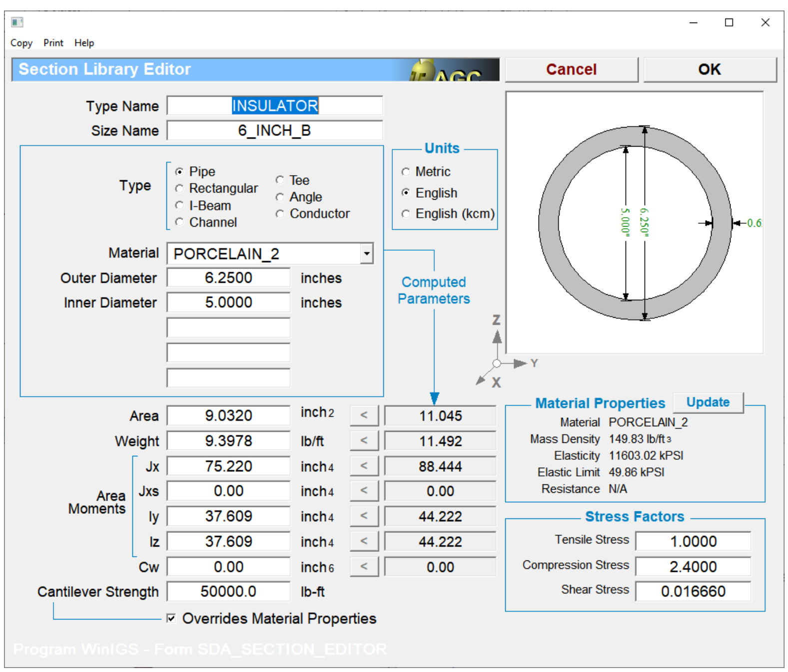


Figure C.3: Insulator Modeling using Cantilever Strength

The manufacturer specified ***maximum permitted bending moment*** (8000 lb-ft) is entered in the Cantilever Strength entry field. Note that the check box titled "***Overrides Material Properties***" is checked so that cantilever strength is used to determine the permitted maximum stress, rather than material library properties.

If the manufacturer provides maximum tensile, compression and torsional strength values, the stress factors can determined and entered in the section properties corresponding fields. First the section geometric parameters must be calculated, as follows:

$$\boldsymbol{A} = \pi(\boldsymbol{r}_2^{\,2} - \boldsymbol{r}_1^{\,2}) = 11.044\,\boldsymbol{in}^2$$
$$\boldsymbol{J} = \pi\frac{\boldsymbol{r}_2^{\,4} - \boldsymbol{r}_1^{\,4}}{2} = 88.4411\,\boldsymbol{in}^4$$
$$\boldsymbol{r} = 6.25''/2 = 3.125''$$

$$\boldsymbol{f}_t = 1$$
$$\boldsymbol{f}_c = \frac{\boldsymbol{F}_{Comp}}{\boldsymbol{F}_{Tens}} = \frac{60{,}000\,\boldsymbol{lb}}{25{,}000\,\boldsymbol{lb}} = 2.40$$
$$\boldsymbol{f}_s = \frac{\boldsymbol{T}_{Tors}\boldsymbol{r}\,/\,\boldsymbol{J}}{\boldsymbol{F}_{Tens}\,/\,\boldsymbol{A}} = \frac{20{,}000\,(\boldsymbol{lb\,in})\times 3.125(\boldsymbol{in})\,/\,88.441(\boldsymbol{in}^4)}{25{,}000\,\boldsymbol{lb}\,/\,11.044(\boldsymbol{in}^2)} = 0.3121$$

The above values are entered in the section stress factors fields. Using the entered data WinIGS sets the maximum permissible stresses, overriding the corresponding material properties values.

Specifically, from the cantilever strength value, WinIGS automatically computes the maximum permissible tension stress as follows:

$$\boldsymbol{S}_{Tcrit} = \frac{2\boldsymbol{T}_{c\max}\boldsymbol{r}}{(\boldsymbol{I}_z + \boldsymbol{I}_y)}$$

where T_{cmax} is the maximum permissible cantilever moment
r is the outside radius of the insulator,
I_z is the moment of inertia about the z axis, and
I_y is the moment of inertia about the y axis.
(the x axis is assumed to be perpendicular to the insulator section)

Using the example insulator parameters:

$$r = 6.25''/2 = 0.07937m$$
$$H = 24'' = 0.609m$$
$$A = \pi(r_2^{\,2} - r_1^{\,2}) = 0.007126m^2$$
$$I_z = I_y = \pi \frac{r_2^{\,4} - r_1^{\,4}}{4} = 0.18406 \times 10^{-4}\ \mathrm{m}^4$$
$$F_{c\max} = 4000lb = 17{,}792N$$

$$S_{Tcrit} = \frac{2F_{c\max}Hr}{(I_z + I_y)} = \frac{17{,}792N \times 0.609m \times 0.07937m}{0.1840 \times 10^{-4}} = 46.78MPa$$

Finally, the maximum permissible tensile, compression, and shear stress values are computed using the user specified stress factors (See Figure C.2 – bottom right side) as follows:

$$S_{T\max} = f_t \times S_{Tcrit}$$
$$S_{C\max} = f_c \times S_{Tcrit}$$
$$S_{S\max} = f_s \times S_{Tcrit}$$

where f_t, f_c *and* f_s are the user specified tensile, compression and shear stress factors, respectively.

C.2: Using the Tensile Strength Specification.

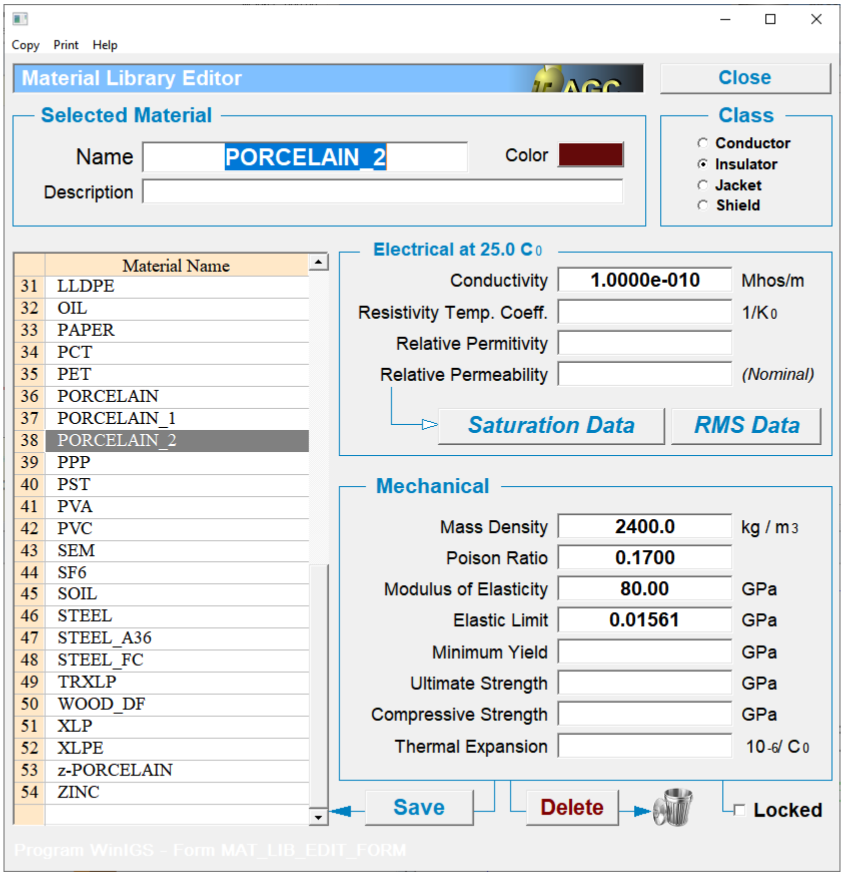
In the example specs shown in Figure C.1, the tensile strength is specified at 25,000 lb. In order to compute the maximum tensile stress, the cross-sectional area is needed. Given the diameter of 6.25 inches and assuming a porcelain thickness of one 1.5 inches the area is computed at

$$A = \pi(r_2^{\,2} - r_1^{\,2}) = 0.007126m^2$$

and the max tensile strength:

$$S_{Tmax} = F\ /\ A = 15.60\ MPa$$

The above value is entered in the material properties for PROCELAIN_2 as illustrated in Figure C.4. In the section properties of the insulator model, the check box titled "Overrides Material Properties" is **un-checked** so that the cantilever strength entry is not used, and the permitted maximum stresses are derived from the material elastic limit.



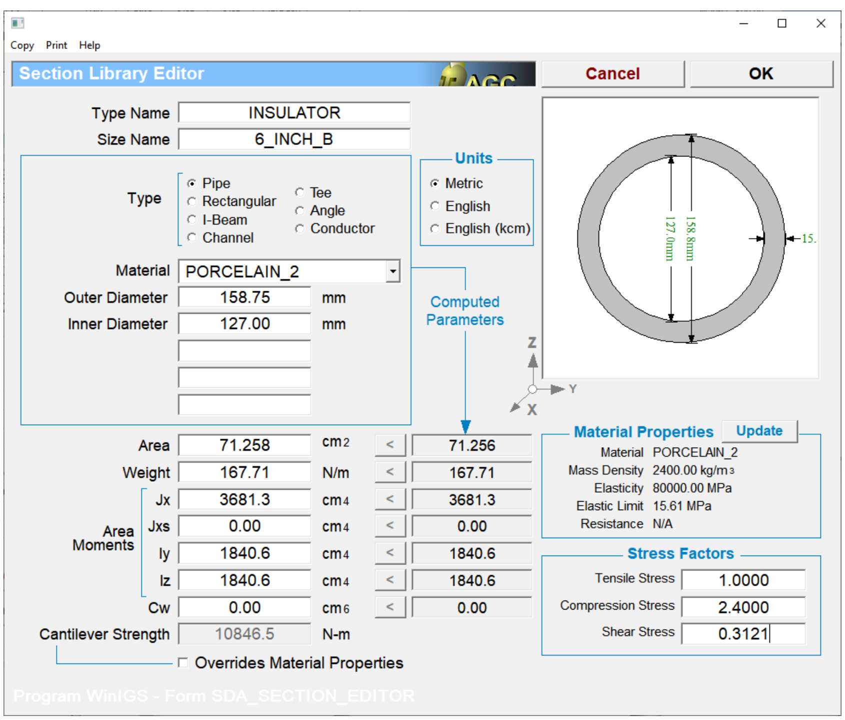
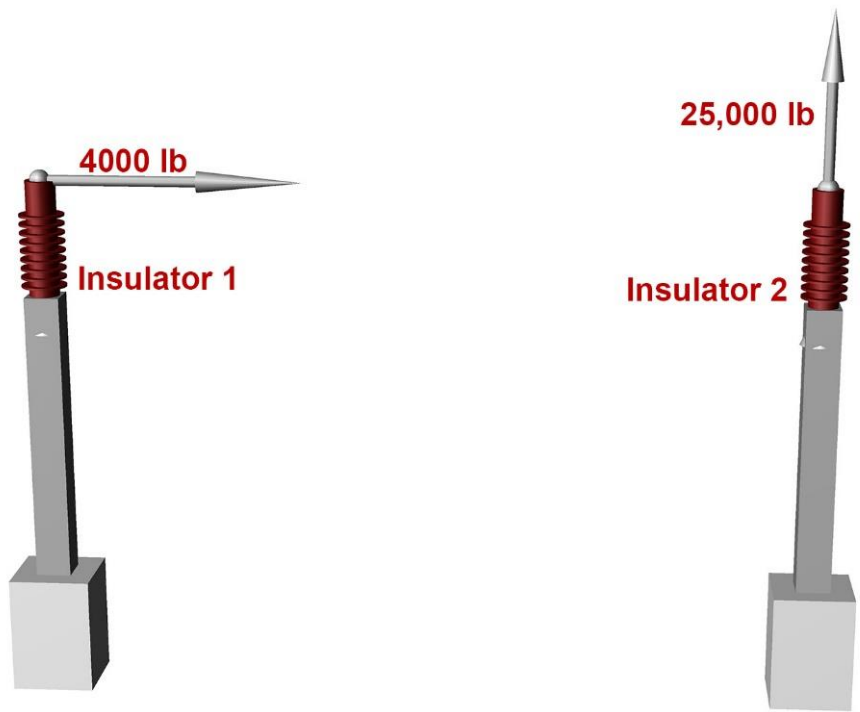


Figure C.4: Insulator Modeling using Specified Tensile Strength

C.3: Insulator Stress Simulation.

In order to verify the insulator model consistency, a simulation was performed containing two insulators, insulator-1 based on the max cantilever strength specification and insulator-2 based on the tensile stress specification. A 1 Hz sinusoidal horizontal force was applied at the top insulator 1 with peak force at 4000 lb, and a sinusoidal vertical force was applied at the top of insulator 2 with peak value of 25,000 lb. The tensile stress at each insulator bottom end was monitored and compared to the max permitted value (see Figures C.5). Plots of the stress versus time are shown in Figure C.6. Note that while the maximum actual stresses on the two insulators are different, each insulator actual maximum stress is approximately equal to the corresponding permissible value.



Location	Element	Tensile Stress	Allowable (Ft)	Overall Margin
1	INSULATOR-1	46.74 MPa	46.77 MPa	0.07 %
2	INSULATOR-2	15.61 MPa	15.61 MPa	0.01 %

Figure C.5: Simulation Model And Maximum Stress Results

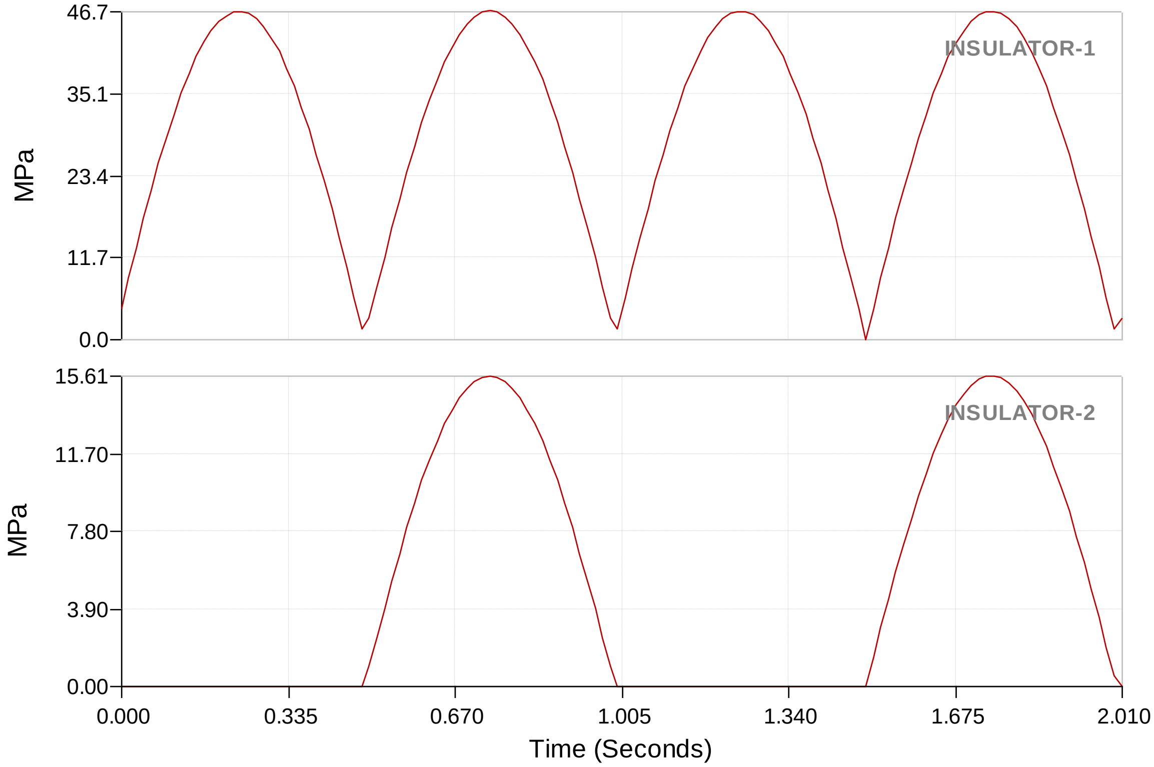


Figure C.6: Plots of Computed Stress vs Time