POWERLINE TECHNOLOGY, INC.

Engineering Software for Utilities

PoleForeman V7 Users Guide

PowerLine Technology, Inc.
P.O. Box 398
Chelsea, AL 35043-0398
205.618.7801
www.powerlinetech.com

Table of Contents

Section

1

Introduction

PoleForeman is a Windows® based software package for use by engineers and technicians who design and maintain overhead utility lines. The program is designed to assist in classing wood utility poles and sizing guy strands to meet the basic strength requirements outlined in the National Electrical Safety Code (NESC). Prevailing practice and most state laws require that utility lines (poles, anchors, guy strands, etc.) be designed to meet the basic requirements of the NESC. PoleForeman provides an efficient and accurate method of determining if a wood structure is in compliance with the basic NESC strength requirements.

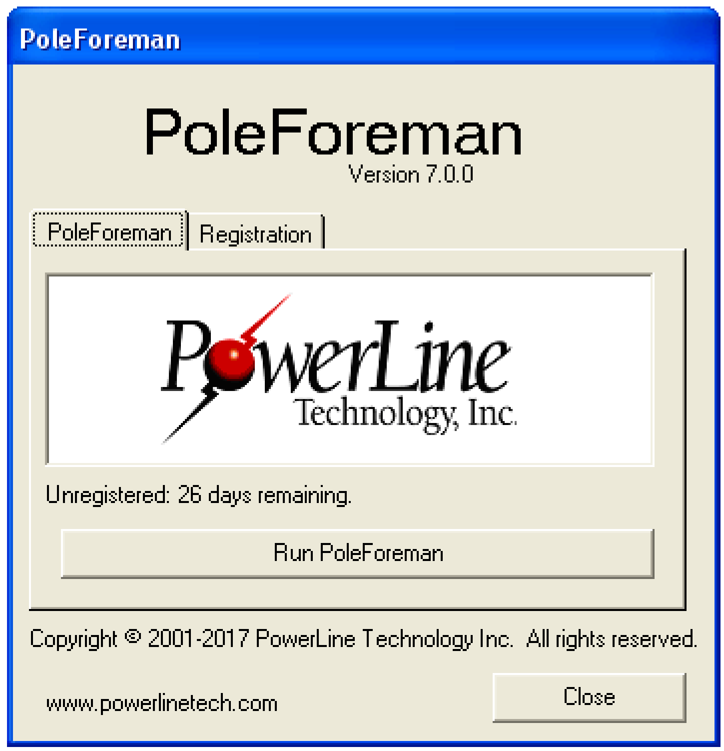
Installation

To install PoleForeman:

1. Insert the PowerLine Technology installation disc in the CD-ROM drive.
2. If you have Autorun enabled, follow the on-screen instructions.
3. If Autorun is not enabled, select Run from the Start Menu.
4. Type D:\setup.exe, click OK, then follow the on-screen instructions (where D: is the CD drive).

Registration

To run PoleForeman, click *Start, Programs, PowerLine Tech* and *PoleForeman*. If the program is unregistered or a trial version, the registration screen is displayed. Click the *Run PoleForeman* command button to start using the program.



Product Support

For product support and technical assistance, send emails to support@powerlinetech.com or give us a call at 205.618.7801.

Section

2

Getting Started

PoleForeman is one of the industry's leading software programs used to class wood poles and calculate guy wire tensions. With today's increased legal liability and the sharing of pole space between multiple companies, utilities are using PoleForeman to help assure compliance with the National Electrical Safety Code.

Modeling Poles

To launch PoleForeman, click *Start, Programs, PowerLine Tech* and *PoleForeman.* If the registration screen is displayed, click *Run PoleForeman.* The program interface is displayed as shown in Figure 1 .

Note

The PoleForeman database contains pre-defined sag tables that specify the conductor tensions used in the pole loading analysis process. In order to obtain accurate analysis results, conductors should be installed in accordance with these pre-defined sag tables. If the conductors are not installed in accordance with these sag tables, then the user must manually enter an appropriate tension for the analysis.

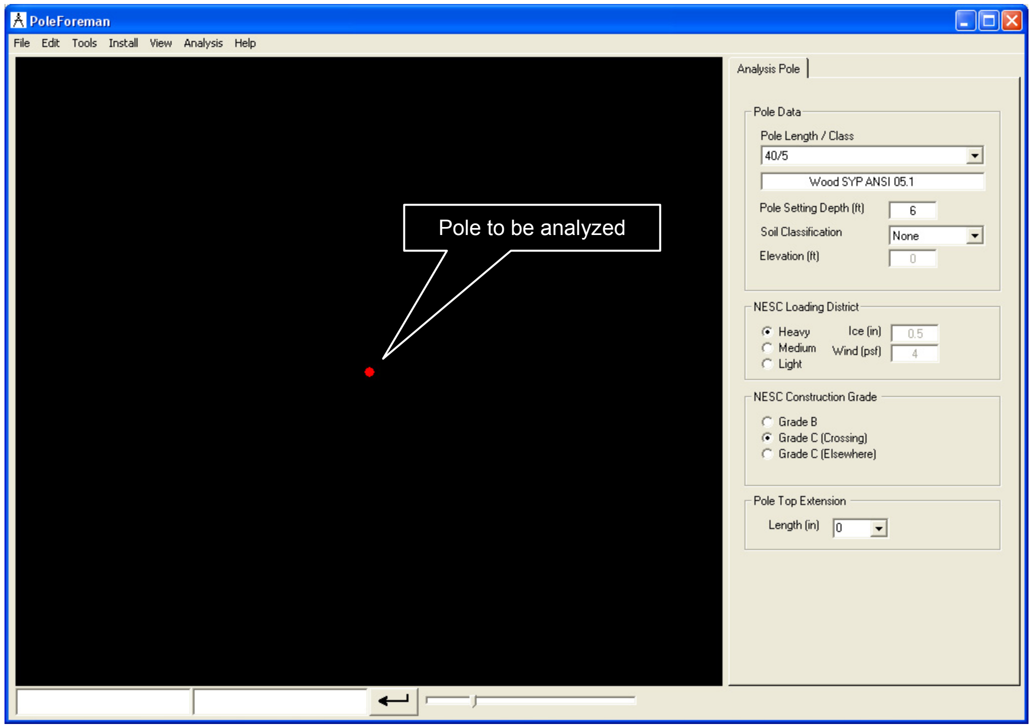


Figure 1 – PoleForeman Interface

The following steps guide you through modeling a simple three phase tangent pole. The process determines if the pole is properly sized to meet the basic strength requirements of the NESC.

Step 1 – Pole Length / Class

The symbol used to represent a pole is a solid circle. The pole to be analyzed is located in the center of the drawing area. Using the mouse, click on the pole in the center of the screen. When selected, the pole is highlighted red and the pole data tab is enabled on the right side of the screen. On the pole data tab, select the size (length/class) of the pole to be analyzed. For this example, select a 40/5 pole.

Step 2 – Pole Setting Depth

The pole setting depth is automatically populated. Edit this field as necessary. For this example, leave the setting depth at 6.0 ft.

Step 3 – Pole Soil Classification

The Pole Soil Classification is used to check if the pole setting depth and selected soil properties provide a stable foundation. Users can select from *"Good", "Average",* or *"Poor"* soil classifications. In no soil classification is selected, the program assumes a rigid foundation and does not check the foundation stability. For additional information and soil classification descriptions, refer to the Appendix. Select *"Good"* for this example.

Step 4 – NESC Loading District

In the NESC loading district frame, select the district in which the pole will be installed. See Figure 250-1 in the NESC. The loading district determines the amount of ice and wind load on the structure. Select *"Medium"*.

Step 5 – NESC Construction Grade

Select the grade of construction for the structure. The construction grade specifies the NESC load factors and strength factors for the structure. See Section 24 of the NESC for additional information about construction grades. Select *"Grade C-Crossing"*.

Step 6 – Install Span

In the menu bar, select *Install*, then *Span*. The program adds a pole and span in the drawing area. You only model the adjacent spans to the pole being analyzed (pole in center of screen). The adjacent poles are considered reference points used to specify span length and direction.

Step 7 – Select Adjacent Pole

Click on the pole that was added in Step 5. The pole turns red when selected. With the pole selected, enter the pole length and setting depth for the adjacent pole. The pole data is entered on the right side of the screen. Select a 40 ft. pole for this example. Leave the default setting depth at 6.0 ft and the pole elevation at 0 ft.

Step 8 – Enter Span Length and Direction

There are three methods that can be used to enter the span length and direction of the adjacent pole added in Step 5. Again, remember that the pole being analyzed is located in the center of the screen. Span lengths and directions are measured relative to the pole being analyzed. The span length and direction is displayed in the lower left portion of the screen. In this example, use one of the following methods to enter a span length of 200ft and a direction of 90°.

- **Mouse Method**
  Using the right mouse button, click on the pole added in Step 5 (adjacent pole). With the right mouse button held down, drag the pole to the desired span length and direction.

- **Arrow Keys Method**
  Using the left mouse button, select the pole added in Step 5. Use the arrow keys on the keyboard to change the span length and direction.

- **Manually Enter Method**
  Using the left mouse button, select the pole added in Step 5. In the lower left portion of the screen, enter the span length and direction for the adjacent pole and hit the <Enter> key on the keyboard to process the changes.

Step 9 –Select Span

Click on the span that was added with the pole in Step 5. When selected, the span is red and the span data tab is displayed on the right side of the screen (Power Conductors, Communication Cables).

Step 10 – Primary Conductor

Select the primary conductor type. For this example, select #2 ACSR.

Step 11 – Primary Conductor Ruling Span

Enter the ruling span for the primary conductors. For this example, enter 200 ft. See Appendix for more information on ruling spans.

Step 12 – Number of Phases

Enter the number of phases in the primary circuit. For this example, select 3.

Step 13 – Neutral Conductor

Select the neutral conductor type. For this example, select #2 ACSR.

Step 14 – Neutral Conductor Ruling Span

Enter the ruling span for the neutral. Enter 200 ft for this example.

Step 15 - Construction Detail

Click on the *Construction Detail* command button. The construction detail screen is used to specify how and where the primary and neutral conductors are supported on the pole. The construction screen is shown in Figure 2.

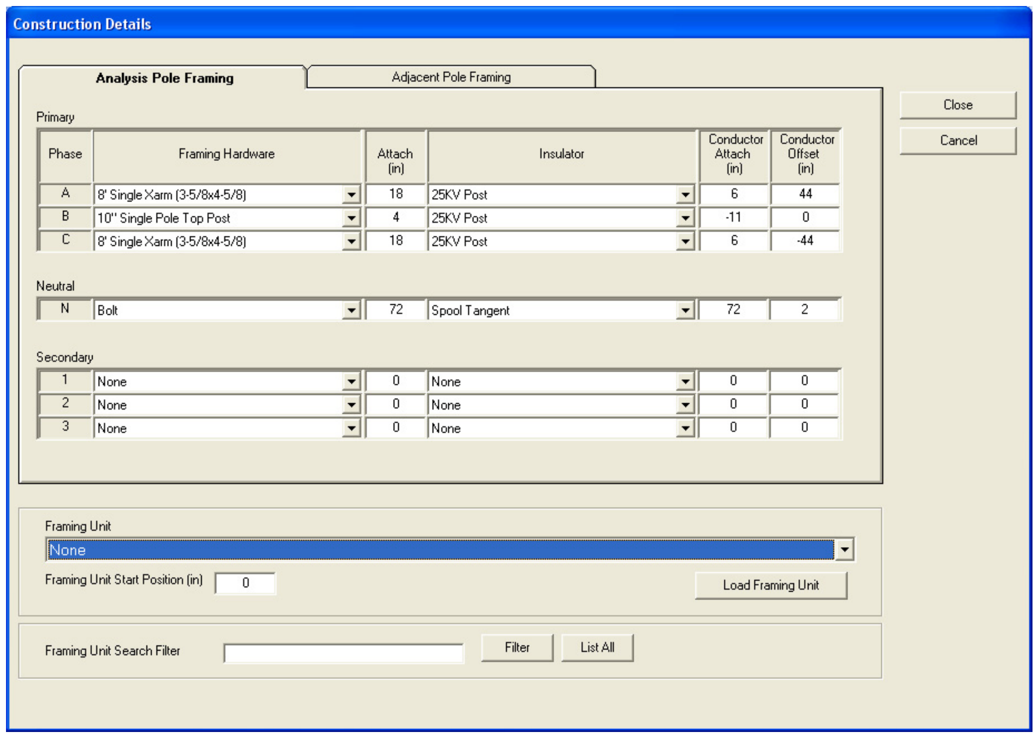


Figure 2 – Construction Screen

Step 16 – Arm/Bracket

On the Construction Details screen, select the arm used to support primary phase A. Select "8' Single Xarm (3-5/8 x 4-5/8)".

Note

Refer to Section 3 to learn about "Framing Units" for a more efficient / simpler method of framing pole top assemblies.

Step 17 – Arm Attachment

Attachments are measured from the pole top to point of attachment in inches. Enter the attachment for the arm selected in Step 15. Enter "18" for this example.

Step 18 – Insulator

Select the type of insulator used to support phase A. Select "25KV Post" from the insulator combo box.

Step 19 – Conductor Attachment

The conductor attachment field is automatically calculated if the arm and insulator data has been entered. If the arms and insulators are excluded from the analysis, enter the conductor attachment in inches. Attachments are measured from the pole top to the point of attachment.

Step 20 – Conductor Offset

Conductor offset is the horizontal distance from the centerline of the pole to the conductor. Enter "44" for phase A conductor offset.

Repeat steps 16-20 for primary phase B, phase C and neutral conductors. Enter the following data.

Phase	Arm / Bracket	Arm Attachment	Insulator	Conductor Attachment	Conductor Offset
B	10" Single Pole Top Post	4	25KV Post	-11	0
C	8' Single Xarm (3-5/8 x 4-5/8)	18	25KV Post	6	-44
N	Bolt	72	Spool Tangent	72	1

Step 21 – Close

Click *Close* on the Construction Details screen.

Step 22 – Copy Span

Copy the span that was created in Steps 5 – 20 and paste in the drawing area. This action adds another adjacent span to the structure. To copy, select the span added in Step 5. The span is highlighted in red. In the menu bar, click *Edit* then *Copy*.

Step 23 – Paste Span

In the menu bar, click *Edit* then *Paste*. A new span is added to the drawing area that contains duplicate conductor information and construction details as the span that was copied. Change the span length and direction of the new span to 200 ft. and 270° respectively (Refer to Step 7 to change span length and direction). The data entry for span 2 is now complete.

With the pole and span data entered, the structure can be viewed as a three dimensional model to verify the correctness of the attachments. A key factor in the accuracy of the analysis is proper attachments for conductors, guy strands, etc.

Step 24 – Solid Model View

In the menu bar, click *View*, then click *Solid Model*. This displays the pole and attachments in a three dimensional view. The solid model screen is shown in Figure 3.

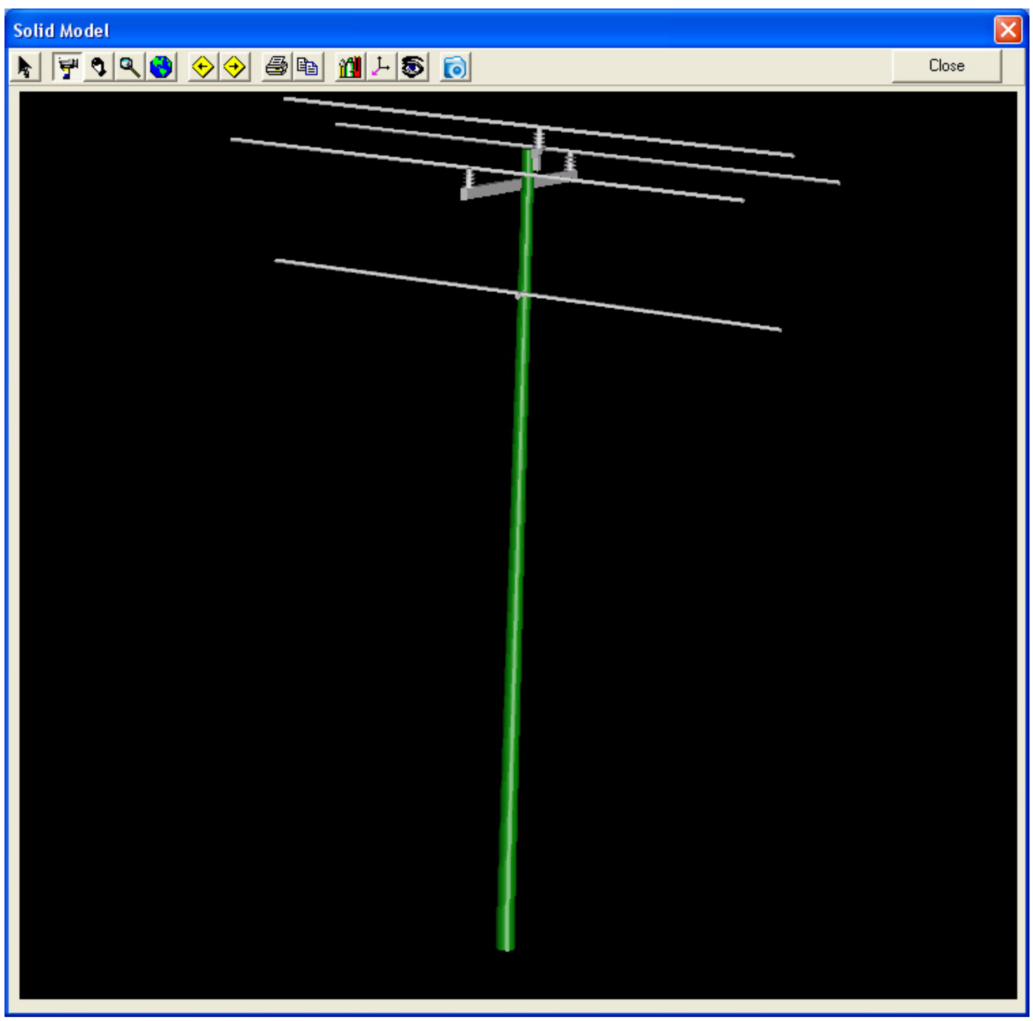
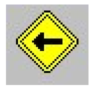


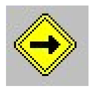
Figure 3 – Solid Model View

Step 25 – Rotate Structure

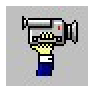
The pole can be viewed from any perspective by using the tools in the toolbar. Click and hold the mouse down on the yellow rotate button in the tool bar to rotate the structure about the vertical axis. The toolbar functions are described below:



Rotate – Rotate structure counter-counter clockwise about vertical axis. – Click and hold mouse down on icon to rotate.

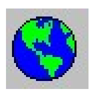


Rotate – Rotate structure clockwise about vertical axis. – Click and hold mouse down on icon to rotate.



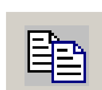
Orbit – Tilt/rotate structure about horizontal and vertical axis. – Click on icon to select orbit tool. Click and hold mouse down on structure. Move mouse to tilt/rotate.

Zoom Window – Zoom in on specific area of structure. – Click on icon to select zoom tool. Use mouse to draw zoom box around area of interest.



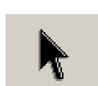
Zoom All – Resets Zoom to full picture. Click on icon to Zoom All.

Pan – Moves structure in graphics area. Click on icon to select Pan tool. Click and hold mouse down on structure. Move structure to desired location. Release mouse.



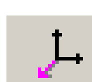
Copy – Copies solid model image to clipboard.

Print – Prints solid model image.

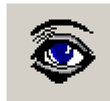


Select – Tool used to select and identify solid model elements such as conductors or cables.

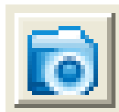
Color – Displays background color palette.



Tilt – Sets default tilt angle for solid model.



Prior View – Orients solid model to viewing perspective of last close event.



Snap Shot – Captures current solid model view and includes that image in the PoleForeman report printout.

The solid model view is used to verify the correctness of conductors, guy strands, and other attachments on the pole. If an attachment does not appear to be correct, close the solid model window. In the topology view, correct the problem by selecting the span, anchor, or other item and then editing the attachment data.

After verifying the solid model to be correct, the structure is ready to be analyzed. Close the Solid Model window by clicking *Close* in the upper right corner of the screen.

Step 26 – Analyze Structure

In the menu bar, click *Analysis* then *Ice and Wind (250B).* The program develops a mathematical model of the structure; determines the forces acting upon it and the reactions of the structure to these forces. Dead loads such as wire tension, pole weight, etc. and live loads such as wind loading are taken into account during the analysis. NESC load factors and material strength factors are also applied during the analysis. The results of the analysis are displayed in the analysis window shown in Figure 4.

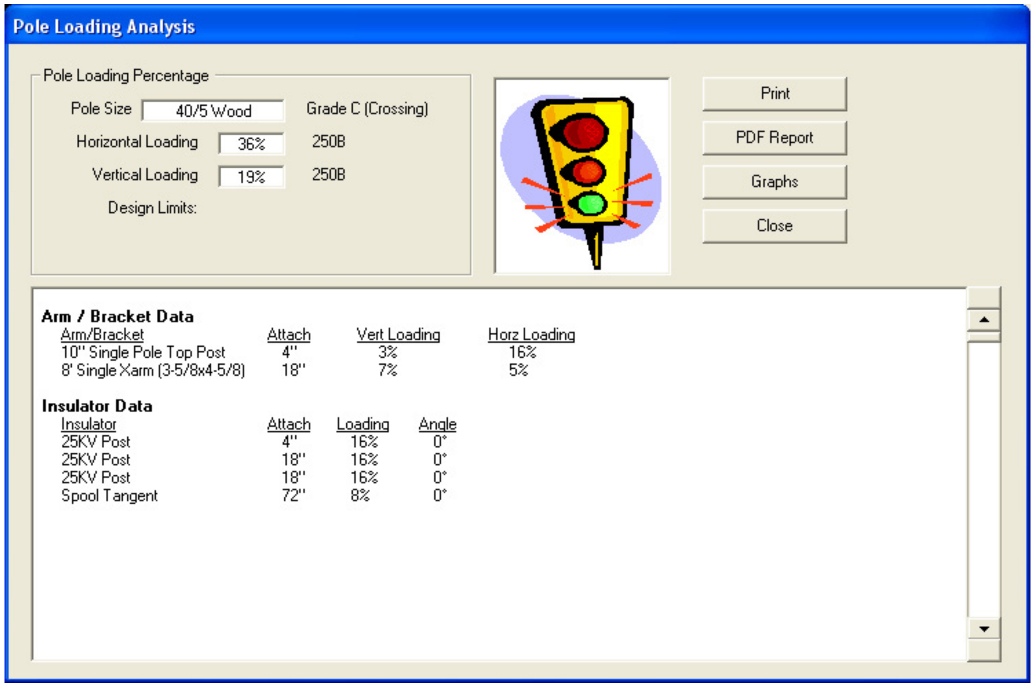


Figure 4 – Analysis Screen

The horizontal loading and vertical loading for the pole are shown as a percentage of allowable pole capacity. Arm and insulator loading is also shown as a percentage of allowable capacity. The green traffic light indicates the pole meets the basic strength requirements of the NESC. In the event a pole is not properly classed or guy strands are not adequately sized to meet basic NESC strength requirements, the program would display a stop sign and highlight the problem area in red.

Note

Although a structure may meet the basic NESC strength requirements, other design considerations could require larger and stronger structures. It is recommended that you review the NESC and determine if additional loading conditions or design criteria should be taken into consideration.

Modeling Anchors

This section describes how to analyze the effects of adding anchors and guy strands to a structure. If properly designed and installed, anchors increase the strength of a structure. In this section, the tangent structure previously modeled will be modified to include an anchor and guy strand. The following steps guide you through installing an anchor.

Step 1 – Modify Tangent Structure

Using the tangent structure previously modeled, select adjacent pole #1 (click on adjacent pole with mouse). The pole is highlighted in red. Using one of the methods described in Step 7 on page 5, change the span direction from 90° to 75°.

Step 2 – Analyze Structure without Anchor

In the menu bar, click *Analysis* then *Ice and Wind (250B)*. The Analysis screen displays a stop sign to indicate that the strength of the structure does not meet the strength requirements of the NESC. The analysis screen is shown in Figure 5.

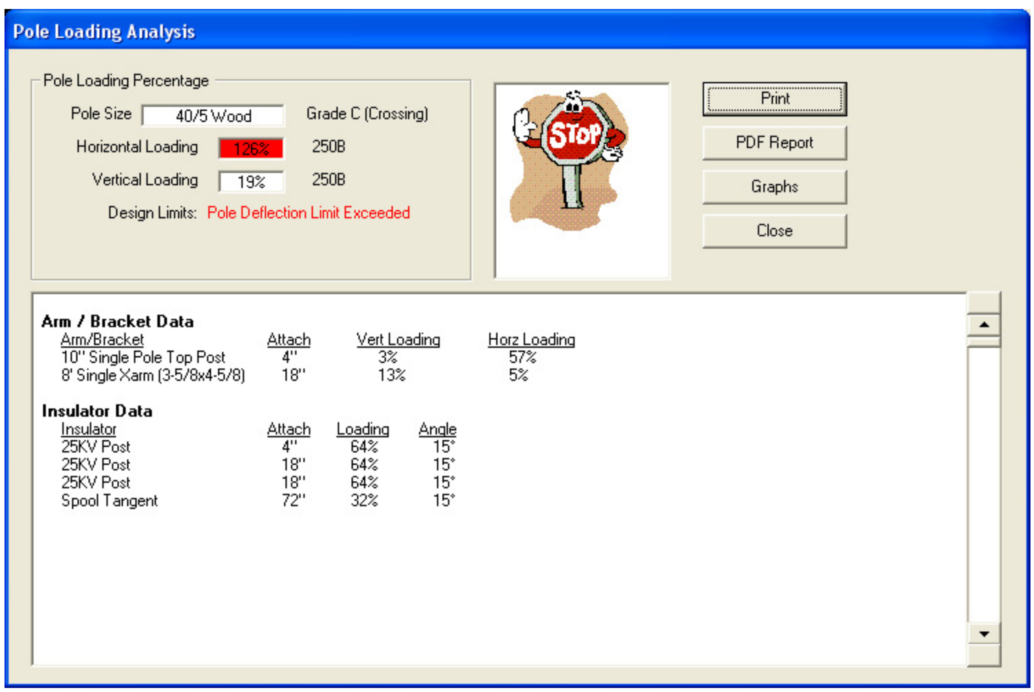


Figure 5 – Analysis Screen

To increase the strength of the structure, an anchor and guy strand will be installed.

Step 3 – Install Anchor

If the Analysis screen is open, click *Close*. In the menu bar, click *Install* then *Anchor*. The symbol for an anchor is added to the analysis pole in the center of the drawing area. The direction of the anchor defaults to 180°. (Note: To have the program automatically place the anchor in the theoretical guy direction, hold down the CTRL key and select the spans to be guyed. In the menu bar, click *Install* then *Anchor*).

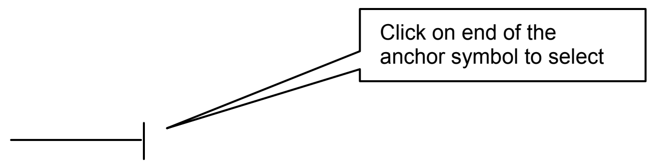


Figure 6 – Anchor Symbol

Step 4 – Select Anchor

Using the left mouse button, click on the end of the anchor symbol. The anchor is highlighted in red and the anchor data tab is displayed on the right side of the screen. The lead length and direction of the anchor is displayed in the lower left portion of the screen.

Step 5 – Lead Length and Direction

Using one of the methods described in Step 7 on page 5, change the lead length and direction of the anchor to 15ft and 172° respectively.

Step 6 – Strand Size and Attachment

On the anchor data tab, select the number of guy strands attached to the anchor rod. Specify the guy strand size and strand attachment for each strand. Select *3/8" SM* for this example. Guy strand attachments are measured from the pole top to the point of attachment in inches. Enter the following attachments:

Strand 1	20"
Strand 2	72"

Step 7 – Anchor Data

On the anchor data tab, select the rod size, anchor type, and soil class. For this example, select a *5/8" Rod*, *8" Single Helix Anchor*, and *Class 4* soil. For a detail description of soil classes, click on the soil class combo box and press F1.

Step 8 – Solid Model View

In the menu bar, click *View* then *Solid Model*. Using the methods previously described, rotate the structure and visually verify that the guy strand attachments are correct. Close Solid Model window.

Step 9 – Analyze Structure

In the menu bar, click *Analysis* then *Ice and Wind (250B)*. The results of the analysis are shown in Figure 7.

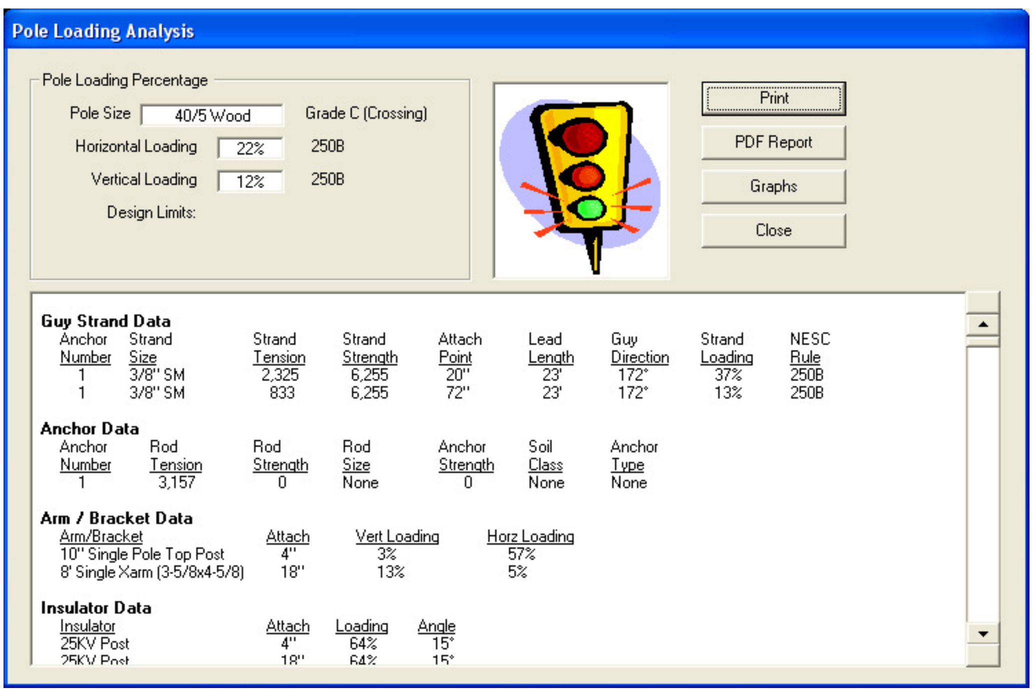


Figure 7 – Analysis Screen with Anchors / Guy Strands

Repeat steps 3 thru 9 to add additional anchors and guy strands to a structure. Each anchor can have up to three guy strands attached to the anchor rod.

Modeling Services

Service conductors increase the mechanical loading on a structure and must be considered when determining a pole's class. The following steps describe how to install services in PoleForeman.

Step 1 – Install Service

In the menu bar, click *Install* then *Service*. The symbol for a service is displayed in the drawing area.

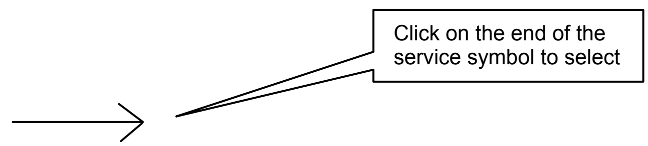


Figure 8 – Service Symbol

Step 2 – Select Service

Using the left mouse button, click on the end of the service symbol. The service is highlighted in red and the service data tab is displayed on the right side of the screen. The length and direction of the service is displayed in the lower left portion of the screen.

Step 3 – Length and Direction

Using one of the methods described in step 7 on page 5, change the length and direction of the service.

Step 4 – Power Service Drop

On the Service drop tab, select the power service conductor type.

Step 5 – Attachments

Enter the attachment point for the service on the pole. Attachments are measured in inches from the pole top.

Step 6 – Telco and CATV

On the service drop tab, click the tabs labeled Telco or CATV to specify the telephone or cable TV service drops.

Modeling Transformers

Pole mounted equipment such as transformers and regulators increase the mechanical loading on a structure and must be considered when determining a pole's class. The following steps describe how to install transformers in PoleForeman.

Step 1 – Install Transformer

In the menu bar click *Install,* then *Transformer*. The symbol for a transformer is displayed in the drawing area.

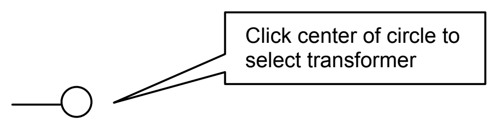


Figure 9 – Pole Mounted Equipment Symbol

Step 2 – Select Transformer

Using the left mouse button, click on the transformer symbol. The transformer will be highlighted in red and the Equipment data tab will be displayed on the right side of the screen. The mounting direction of the transformer will be displayed in the lower left portion of the screen.

Step 3 –Direction

Using one of the methods described in Step 7 on page 5, change the mounting direction of the transformer.

Step 4 – Transformer

On the Equipment data tab, select the transformer size and attachment point. Attachments are measured from the pole top to the top mounting bracket of the transformer in inches.

User Defined Equipment Objects

The user defined equipment object models equipment attachments that are not pre-defined. These items may include banners, antennas, etc. To install a user defined equipment object, follow the steps listed below.

Step 1 – Install User Defined Equipment Object

In the menu bar, click *Tools,* and then click *Equipment Object*. An equipment symbol is added to the pole being analyzed.

Step 2 – Select Equipment Object

If the equipment object symbol is not currently selected (highlighted in red), then click on the center of the equipment symbol.

Step 3 – Specify Parameters

On the right hand side of the screen, the equipment data tab is displayed. Select "User Defined Equipment" from the drop down list. Enter the equipment attachment point. Specify whether the equipment is *Rectangular* or *Cylindrical*. Enter the dimensions and weight for the object.

Step 4 – Enter Mounting Direction

With the equipment object selected, use the mouse or arrow keys to specify the direction the object is mounted on the pole.

Joint Use Cables

To analyze the impact of joint use cables on a structure, follow the steps listed below.

Step 1 – Select Span

On the Topology screen, click on the span in which the joint use cable is to be added. The span is highlighted in red when selected. When the span is selected, the conductor and joint use data tabs are displayed on the right side of the screen.

Step 2 – Joint Use Data Tab

Click on the data tab labeled *"Joint Use"* on the right side of the screen.

Step 3 – Number of Joint Use Cables

Select the number of joint use cables to be installed in the span and then click *Cable Data*.

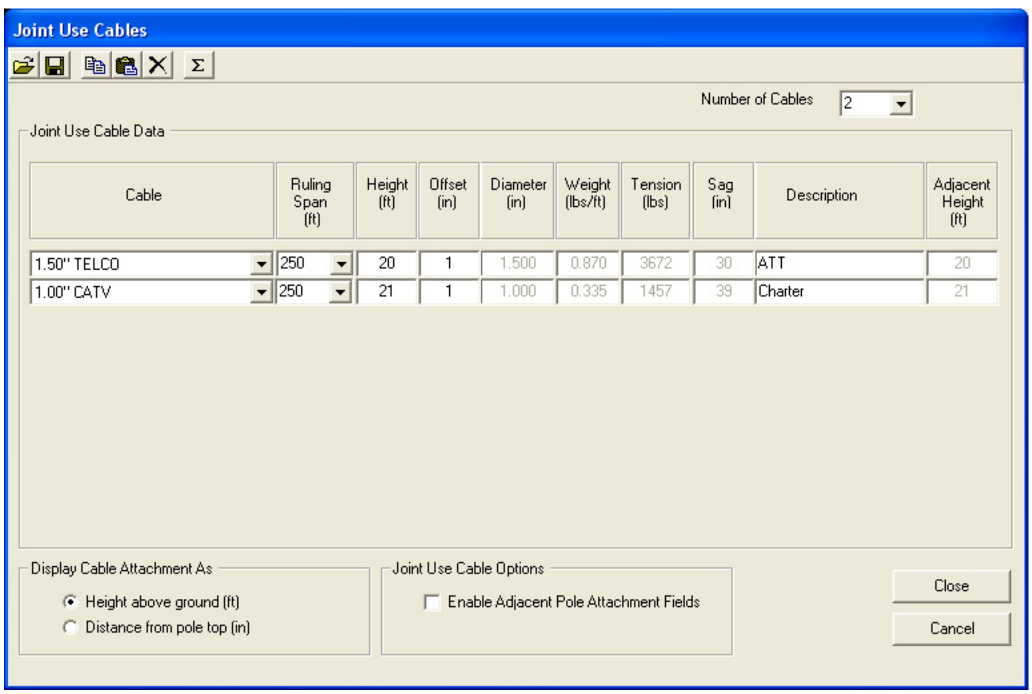


Figure 10 – Joint Use Data Screen

Step 4 – Enter Joint Use Cable Data

The Joint Use Cable Data Screen is shown in Figure 10. The database comes pre-loaded with generic cable data that can be used to model joint use cables in the event no other data is available. It is recommended that cable parameters (diameter, weight, tension, etc.) be provided by the cable owner in order to obtain accurate information. The generic data in the database is provided as a "last resort" for cases where the information needed to perform an analysis is not readily available.

When the parameters for a cable are known, select *User Defined* from the cable combo box. Enter the cable data directly into the Joint Use Cable Data table. The following parameters are required for each joint use cable within the span.

- **Diameter** – Cable diameter. If multiple cables are lashed to a strand, the diameter should be the overall projected diameter of the cable assembly. The diameter is measured in inches.

- **Weight** – Weight per foot of joint use cables and strand. Units – *lbs/ft.*

- **Tension** – The design tension for the cable / cable assembly at the NESC Loading District temperature and loading conditions. Units – *lbs.*

- **Sag** – Sag at the NESC Loading District temperature with no ice or wind. Units – *in.*

- **Attachment/Height** – Users can enter the attachment point of the cable on the pole as either a distance from pole top or as a height above ground line. Select the preferred option for entering cable attachments on the bottom left corner of the joint use screen. If the Height above Ground option is selected, measurements are in feet. The distance above ground will remain constant with changes in pole height and pole setting depth. If the Distance from Pole Top option is selected, measurements are in inches. The distance from the pole top to the cable will remain constant with changes in pole length and setting depth.

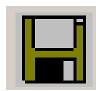
- **Offset** – Horizontal offset of cable from face of pole. Units – *in.*
- **Description** – Name assigned to cable.

If the information described above is not available, select one of the generic cables from the cable combo box. Select a cable that best matches the joint use cable in the field. Enter a ruling span for the cable. The cable diameter, weight, tension, and sag is populated from the database. The sag and tension data for the cable selected is taken from pre-defined sag tables that have assumed installation conditions.

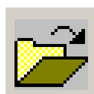
Note

The assumed installation conditions used to create the joint use sag tables may not reflect the actual installation conditions of the joint use cable in the field. It is recommended that the cable owner provide the sag and tension data for their cables.

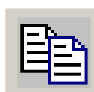
The toolbar on the joint use cable screen provides additional functionality to facilitate modeling cables. Each toolbar button is described below:



Save – Saves cable from active row in data table to a unique file.

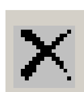


Open – Opens a previously saved cable and inserts that cable into the active row of the data table.

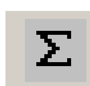


Copy – Copies the cable parameters from the active row in the data table to the clipboard.

Paste – Pastes the cable parameters from the clipboard to the active row in the data table.



Delete – Removes the cable parameters from the active row in the data table.



Estimate Sag and Tension – Loads the cable sag and tension estimation tool. Refer to the section in the appendix on Estimating cable Sag and Tension for instruction on estimating cable tensions.

Manually Specifying Conductor Tensions

To manually specify conductor tensions, follow these steps.

Step 1 – Select Span

Select a span. Click the Primary Data tab. At the bottom of the Primary Data tab, select the check box labeled *Manually Specify Sag and Tension.*

Step 2 – Enter Sag and Tension

Under the Primary Conductor combo box, enter the conductor sag and tension. The tension value entered should be the design tension for the conductor at the appropriate ice, wind, and temperature loadings specified in Table 1 (NESC Rule 250B only). It is recommended that you consult with your company's conductor expert or consultant to assist in deriving the design tension.

Table 1 – Rule 250B Ice, Wind, and Temperature Loadings

NESC LOADING DISTRICT	Temperature (°F)	Wind Loading (lbs/ft^2)	Ice Loading (in)	NESC Constant (lbs/ft)
HEAVY	0°	4	0.50	0.30
MEDIUM	15°	4	0.25	0.20
LIGHT	30°	9	0.00	0.05

The sag value entered should be the sag at the loading district temperature (see Table 1) with 0.00 in. of ice and 0.00 p.s.f. of wind loading.

Examples where manually specified tensions could be required include slack spans or modeling other utilities' conductors on a structure. This feature is available for primary, neutral, secondary, and service conductors.

Note

The loading conditions in Table 1 are for Rule 250B only. Refer to the NESC for the conductor loading conditions for Rule 250C (Extreme Wind) and Rule 250D (Extreme Ice). The loading conditions are different for each rule.

Modeling Pole Top Extensions

Pole top extensions are sometimes used to increase the height of an existing pole. The effects of adding a pole top extension can be modeled in PoleForeman by specify the length of the extension and any new attachments. When a pole top extension is added, the top of the extension becomes the new reference point for all attachments. To model a pole top extension, follow the steps outline below.

Step 1 – Select Pole

On the Topology Screen, select the pole in the center of the screen (Analysis Pole). When selected, the pole will turn red and the pole data tab will be enabled on the right side of the screen.

Step 2 – Specify Pole Top Extension Length for Analysis Pole

On the pole data tab, select the length of the pole top extension from the drop down list. Or, type in the pole top extension length in the drop down box. Pole top lengths are measured in inches. Any existing attachments on the pole are atomically adjusted so that they remain at the same position (attachment point).

Step 3 – Specify Pole Top Extension Length for Adjacent Poles

Select the adjacent poles one at a time and enter the pole top extension length for each adjacent pole. If an adjacent pole does not have a pole top extension, the user will need to verify and/or correct the attachments on the adjacent pole. By default, the program assumes attachment points are the same on both ends of a span.

Step 4 – Add additional circuits and/or equipment

Add any additional circuits or equipment that will be installed on the structure.

Step 5 – Analyze Structure

Click *Analysis* then *Analyze Structure* from the menu bar.

Note

Pole top extensions are modeled in the software as a simple extension to the top of the pole. That is, the program adds the length of the extension to the length of the existing pole to create a new structure. The mathematical model of the new structure is based on the diameters and strengths of the existing pole. The pole top extension is assigned a constant diameter equal to the ANSI 05.1 tip diameter of the existing pole. The existing pole is assumed to meet the dimensions and strengths specified in ANSI 05.1. If the existing pole has any deterioration due to rot or other damage, appropriate action should be taken to evaluate the remaining strength. After assessing the strength, the designer should verify that the calculated loading on the pole does not exceed the assessed strength of the pole.

Section

3

Tool and Tips

Listed below is a brief description of some of the Tools available in PoleForeman.

Templates

Templates are pre-built designs that minimize the time required to analyze a structure. Templates are based on common construction practices published in your company's standards manual. To open a template, click *File* then *Template* in the menu bar. Select the template that matches the structure to be analyzed. Change the NESC Loading District, Construction Grade, pole sizes, span lengths, directions, wire sizes, ruling spans, etc. to reflect the conditions associated with the structure of interest. View the Solid Model. Analyze the structure. The conductor attachments, insulators, arms, etc., are pre-defined in the template.

To save a structure you have created as a template, click *File* then *File Save* in the menu bar. On the file save screen, change the file type to PoleForeman Template (*.pft). Change the *Save In* folder to …\Program Files\PowerLine Tech\Plates. Click *Save*.

Framing Units

Just as templates, framing units expedite the modeling process. Framing units allow you to add additional framing configurations or modify existing configurations without having to manually enter all the data. For example, if you open a template for a 3Ø tangent line and need to add a 3Ø tap pulling off the structure, you simply install the new span and specify the conductor size. Then, on the Construction Details screen, select the framing unit for the 3Ø tap (3Ø Dead End). The framing unit selection box is located near the bottom of the Construction Details Screen as shown in Figure 3-1.

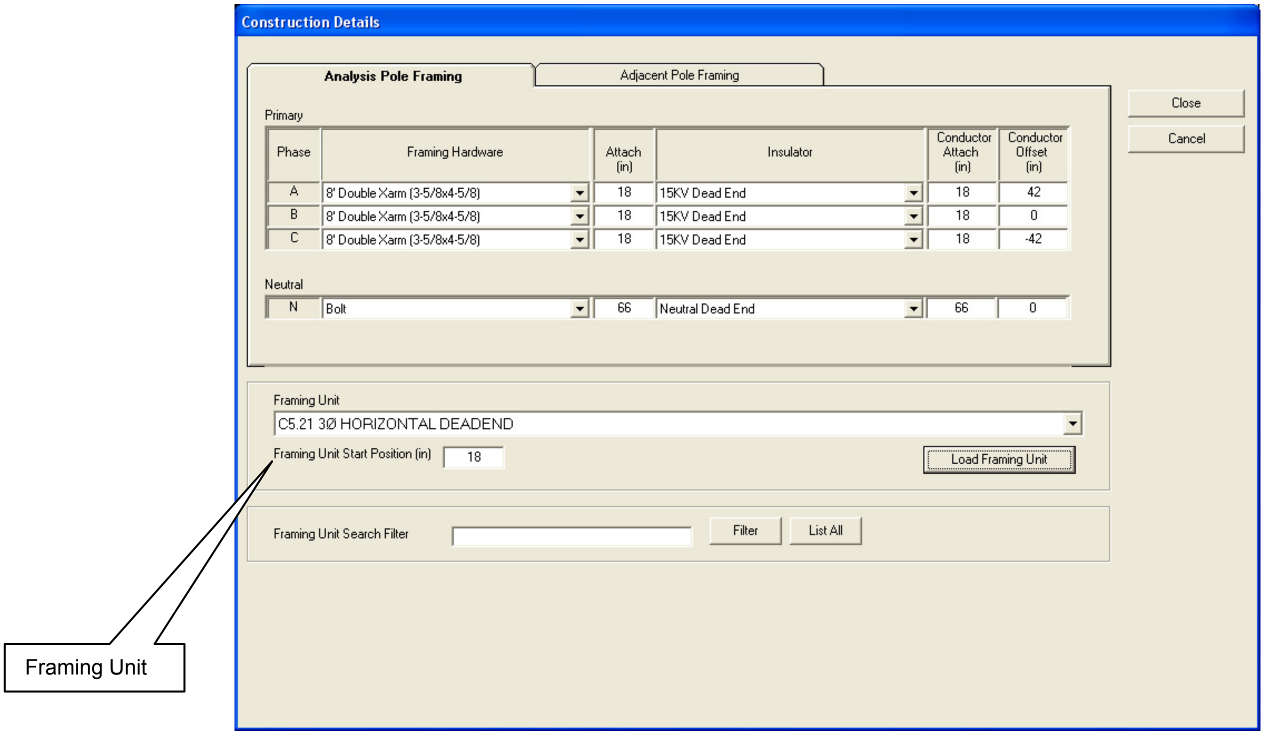


Figure 3-1 Construction Details / Framing Units

After selecting the appropriate framing unit, specify the framing unit start position and click *Load Framing Unit*. The bracket, insulator, attachment and offset data will automatically be populated.

There are three types of framing units: primary and neutral units, primary-only units, and neutral-only units. If a primary-only unit is selected, only the primary framing will be updated. If a primary and neutral unit is selected, both the primary and neutral data will be updated. If a neutral-only unit is selected, only the neutral framing will be updated.

The framing unit start position is the distance from the top of the pole to the starting position of the framing unit (framing unit origin). The start position of the framing unit will vary based on the unit type and how the unit was configured when built. Figure 3-2 depicts a typical framing unit and shows the default start position and framing unit origin.

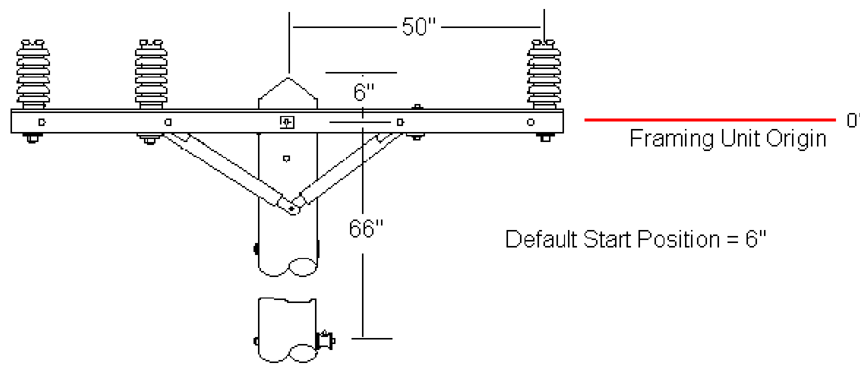


Figure 3-2 Framing Unit Construction Drawing

Captions

The Caption tool adds a caption box to the drawing area. To add a caption, click *Tools* then *Caption* in the menu bar. Double click on the caption to edit. Use the mouse to drag and drop the caption anywhere in drawing area.

Ruling Span Calculator

PoleForeman includes a ruling span calculator available under the Tools menu. Enter the span lengths between dead ends into the ruling span calculator to calculate the ruling span for a section of line.

Guy Direction Calculator

The guy direction calculator determines the theoretical direction to install an anchor. Power conductors and joint use cables are calculated independently of each other. The theoretical direction is based on conductor and cable tensions only. Wind loading is not considered with the guy direction calculator. The guy direction calculator sums forces to determine a resultant horizontal force and provides a guy direction that is in the opposite direction of the resultant force. To use the guy direction calculator, hold down the CTRL key and click on the spans to be guyed. The spans will be highlighted in green. Click *Tools* in the menu bar, then *Guy Direction*. The theoretical guy direction will be displayed.

Note

The theoretical guy direction may not be practical due to field conditions, construction practices, and multiple circuits/spans being constructed at different heights on the pole.

Rotate Drawing

The *Rotate Drawing* feature will rotate all objects in the Topology drawing area by a user specified angle in degrees.

Multiple Circuit Construction

PoleForeman accommodates up to four primary circuits in a span. Circuit 1 is the default circuit. To add an additional circuit to a span, select a span in the drawing area, then use the circuit drop down box located on the Primary Conductor data tab to change the circuit number. Enter the conductor information for the new circuit.

Note

A common practice in multiple circuit construction is for both primary circuits to share a common neutral. In this case, do not specify a neutral conductor for the second circuit.

Save Settings

This feature will save the current program settings as the default settings.

Restore Default Settings

This feature restores the original default settings for the program.

Notes

The *Notes* feature is located under the *File* menu. The *Notes* option allows for the input of any notes or comments pertaining to the file. When an analysis report is created, the contents of the notes module are printed on the report.

Tips when using PoleForeman

- To select multiple spans, select the first span, hold down the shift key, select the second span. Enter the span information on the right side of the screen. The span data is saved for both spans.

- Use the left mouse button to select an object without moving it. For example, once an anchor has been properly placed in the drawing area, click on the anchor with the left mouse button to select. This will prevent accidental changes in length and direction. Use the right mouse button when changing lengths and directions with the mouse.

- Use the Drawing Scale scroll bar (bottom of topology screen) to change the drawing scale. This feature is useful when working with short anchor lead lengths or long spans.

- After selecting an object (pole, service, anchor, etc.), change the length or direction of the object by entering the data in the length and direction text boxes at the bottom of the screen. Press <Enter> after specifying the length and direction to process changes. Use the "+" symbol and "-" symbol to add and subtract degrees from the direction of an object. For example, to add 15° to the direction of an existing span, select the span, enter "+15" in the direction box and then press <Enter>.

- To enter conductor attachments based on conductor spacing, use the "+" symbol as a prefix for the conductor spacing. Example: Enter "8" for primary phase A. Enter "+40" for Primary phase B. Enter "+40" for Primary Phase C. The attachment values displayed will be Phase A: 8", Phase B: 48", Phase C: 88".

- To have the program automatically place an anchor, hold down the CTRL key and click on the spans to be guyed. The spans are highlighted in green. Click *Install* then *Anchor* in the menu bar. The program automatically places an anchor based on the tensions of the power conductors within each span selected. Joint use cable tensions are not taken into consideration when using this feature.

Section 4

Options Screen

The Options screen is used to specify parameters associated with the program. To access the Options screen, click *Tools* then *Options* in the menu bar.

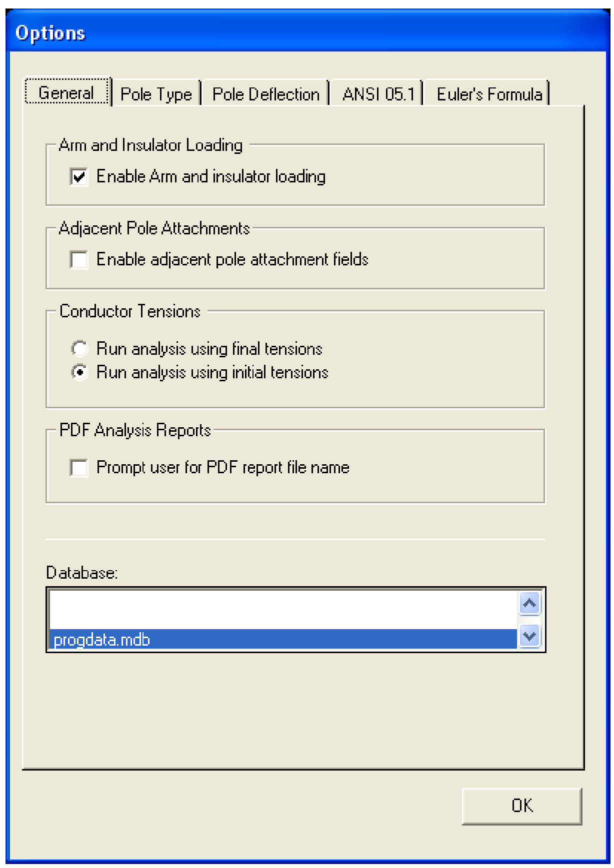


Figure 11 – Options Screen, General Tab

Arm and Insulator Loading

The Arm and Insulator Loading option allows you to turn on and off the arm and insulator loading.

Adjacent Pole Attachments

The program assumes attachments of conductors and cables are the same on adjacent poles. To enter different attachments on adjacent poles, check the *Adjacent Pole Attachments* check box. The adjacent pole attachment fields will be displayed when the span is selected.

Conductor Tensions

The conductor tension option allows the user to specify whether initial or final conductor tensions are used in the pole loading calculations.

PDF Analysis Report

The prompt user for pdf report file name option allows the user to specify if the program prompts for a pdf report file name or if the program auto assigns the pdf report filename with no user prompt.

Pole Type

Pole loading and guying calculations in PoleForeman are based on wood pole dimensions and strengths as specified in ANSI 05.1-2008. The NESC makes allowances for the use of alternate pole material types such as pre-stressed concrete or metal. If an alternate pole material type is used, the alternate material pole must be equivalent in strength to a wood pole of the same length and class. Alternate material types can be selected by clicking *Tools* then *Options* in the menu bar. On the Options screen, click the *Pole Type* tab.

The default material type is wood. When other material types are specified, PoleForeman selects the appropriate load factors and material strength factors from the NESC and applies those factors during the pole loading calculations. As noted above, when alternate material types are specified, the calculations in PoleForeman are still based on wood pole dimensions and strengths. The only changes are in the NESC factors utilized in the analysis. Thus, the requirement that an alternate material pole be equivalent in strength to a wood pole of the same length and class is essential for the results of the analysis to be valid. It is recommended that you consult with your standards engineer when determining the proper class for alternate material types. Refer to the Appendix for information on wood pole equivalencies.

The shape of the pole determines the NESC shape factor applied during the loading calculations. For round or cylindrical poles, a shape factor of 1.0 is applied. For rectangular poles or poles with flat surfaces, a shape factor of 1.6 is applied.

Wood Pole Deflection

When designing overhead power lines, an important aspect to consider is pole deflection. Common practice for limiting pole deflection is to install down guys. When conditions occur that prevent poles from being guyed properly, the designer should take into consideration pole deflection.

On the *Pole Deflection* tab, the user can limit pole deflection to a discrete value or limit deflection based on a percentage of pole height above ground line.

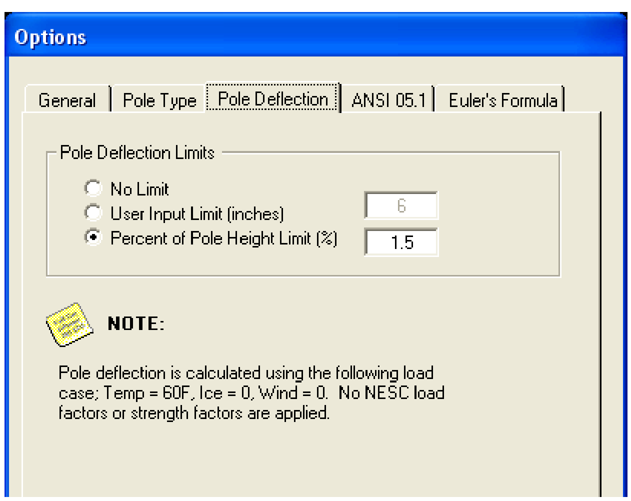
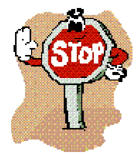


Figure 12 – Pole Deflection Tab

The pole deflection limit is checked under the following load case:

Temp = 60˚F
Wind = 0.00 lbs/ft^2
Ice = 0.00 inches

No NESC load factors or strength factors are applied during the pole deflection analysis. In the event the pole deflection limits are exceeded, the stop sign graphic is displayed along with the message: "Design Limits: Pole Deflection Limit Exceeded"



The NESC does not have defined limits for pole deflection and therefore the deflection limits are left to the discretion of the designer. If the designer elects to exclude pole deflection limits from the analysis, simply select *No Limits* on the *Pole Deflection* tab.

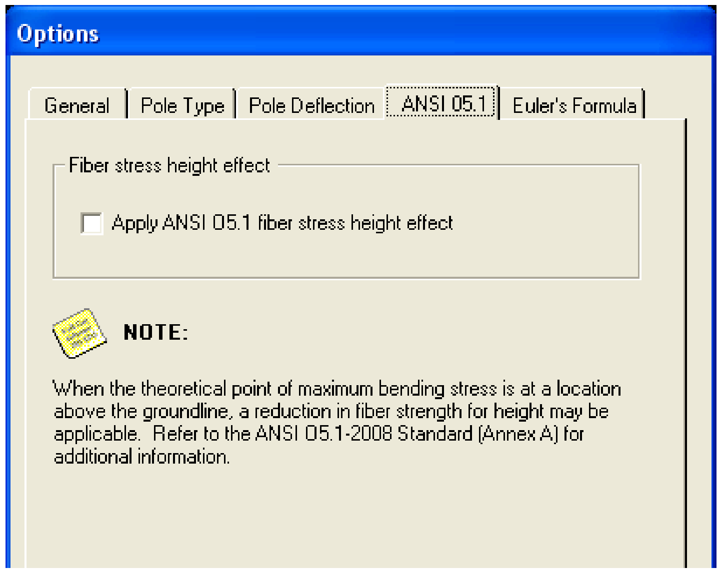
Note

1. The pole loading and pole deflection calculations in the PoleForeman software assume the pole is installed in a rigid foundation. The effects of poor soil conditions and/or inadequate foundations are not taken into consideration during the analysis process.

2. When conductor tensions are manually specified, the manually specified tension will be used for all analysis cases. Therefore, the user should enter the tension at 60˚F, 0 ice, 0 Wind, for deflection analysis. Then enter the NESC loaded tension or design tension when performing pole loading and guying analysis.

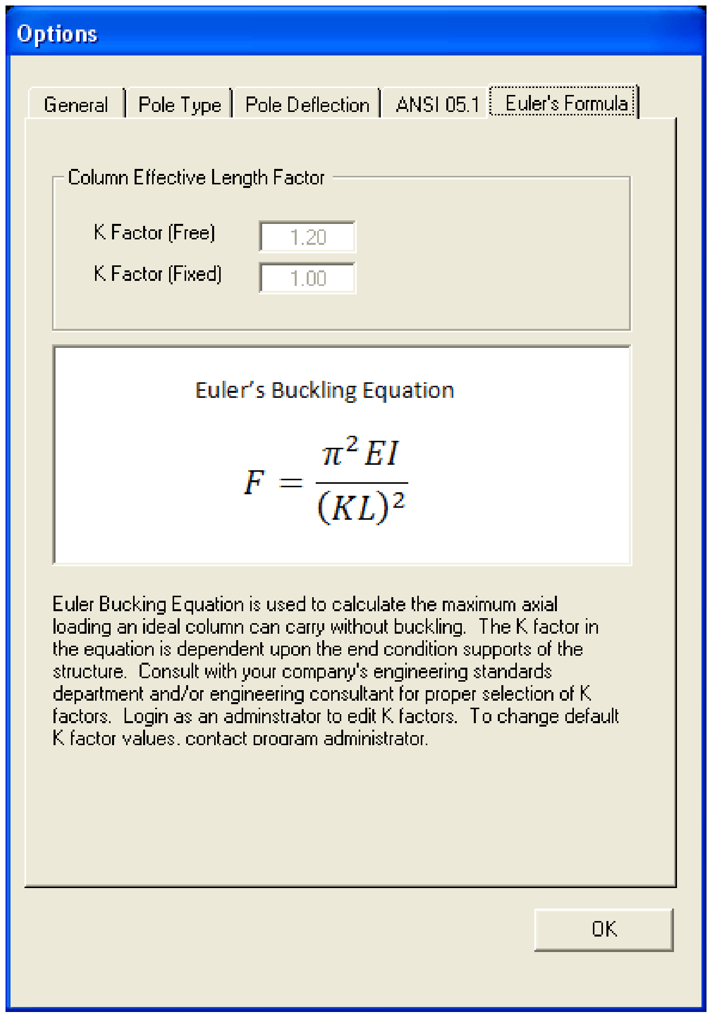
ANSI Fiber Stress Height Effect

To turn on the ANSI 05.1 Fiber Stress Height Effect, select the check box on the ANSI 05.1 Options Tab. Selecting the check box turns on the height effect for all poles. Refer to the Annex A in ANSI 05.1 – 2008 for addition information.



Euler's Buckling – K Factor

Euler Bucking Equation is used to calculate the maximum axial loading an ideal column can carry without buckling. The K factor in the equation is dependent upon the end condition supports of the structure.



To unlock the K Factor input fields in the program, Click *Tools* in the menu bar, and then click *Admin*. Enter your username and "PoleForeman" for the password. After logging in as an administrator, click *Tools* then *Options*. Select the *Euler Equation* Tab. The K factor fields are now unlocked. To change the default K factor values, send email request to *support@powerlinetech.com*.

Section
5

NESC Screen

The NESC screen displays the load factors and material strength factors that are used during the analysis process. These factors add a margin of safety to the strength of the structure as defined in the NESC.

To view the NESC screen, click *Tools* then *NESC* in the menu bar. The NESC screen has separate tabs for the Pole load factors, Guy load factors, Bracket load factors, and Insulator load factors.

The NESC screen is shown in Figure 13.

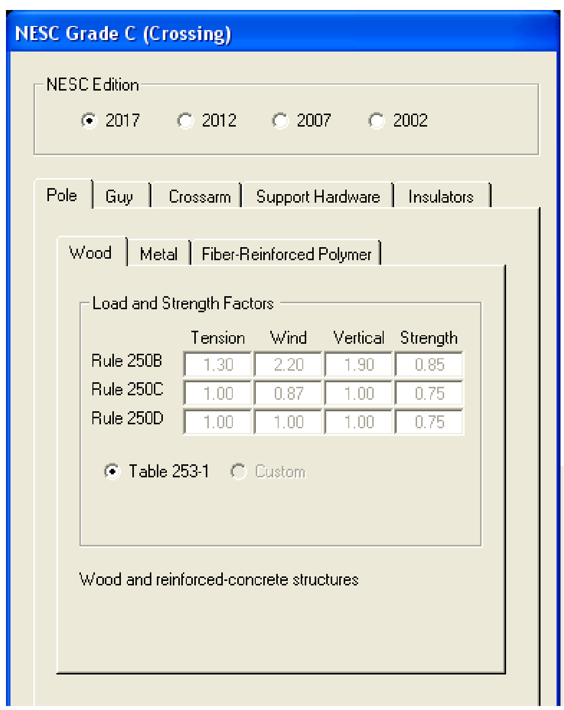


Figure 13 – NESC Screen

Refer to NESC Sections 25 and 26 for additional information on load and strength factors.

To modify the load factors to values other than those specified in the NESC, follow these steps:

1. If open, close the NESC Screen.
2. Click *Tools* then *Admin* in the menu bar.
3. On the Admin Login Screen, enter your user name and then "PoleForeman" as the password. Click OK.
4. Click *Tools* then *NESC* in the menu bar.
5. On the NESC Screen, click the *Custom* radio button.
6. Modify the load factors and strength factors as necessary.

Section

6

Appendix

Ruling Span Theory

Sag and tension data is based upon the assumption that the tension in the conductor will be the same in each span throughout a series of spans of varying lengths between dead-ends. To predict the overall changes in sag and tension in a series of spans, calculations are made using a theoretical span called a ruling span.

The ruling span is calculated by using the following equation:

$$L_{RS} = \sqrt{\frac{L_1^3 + L_2^3 + L_3^3 + ... + L_n^3}{L_1 + L_2 + L_3 + ... + L_n}}$$

Where:

L_{RS} = Ruling Span (ft)

L_1, L_2,…. L_n = Length of each span between dead ends (ft)

Each time a conductor is dead-ended, a new ruling span must be calculated. Proper selection of a ruling span provides a span length representative of the spans in a section of line. Use of an incorrect ruling span in design or construction may result in either excessive tension or excessive sag.

Wood Pole Equivalency

Wood pole equivalency is a term used often, especially when referring to metal and concrete poles. As the term implies, it is used to correlate metal and concrete poles to wood poles. For example, a manufacturer may state that a 40/5 metal pole is equivalent to a 40/5 wood pole. From this statement, it is inferred that both poles have the same load carrying capability or strength. This may or may not be the case.

Some manufacturers of concrete and metal poles have modified the wood pole load carrying capacities in ANSI 05.1 by a ratio of overload factors to arrive at an equivalent tip load for steel or concrete poles. The ratio of wind overload factors is used for this calculation because it is assumed that wind has the greatest impact on the design of a tangent distribution pole.

The ratio that is used is 2.5/4.0. Table A-1 shows the load carrying capability of wood poles as defined in ASNI 05.1 and the load carrying capability of manufactured poles after the ratio has been applied.

Table A-1: Load Carrying Capability

Pole Class	ANSI 05.1 Load Carrying Capability* of Wood Poles	Load Carrying Capability* of Metal and Concrete Poles after Applying Overload Factor Ratio
5	1900 lbs	1188 lbs
4	2400 lbs	1500 lbs
3	3000 lbs	1875 lbs
2	3700 lbs	2313 lbs
1	4500 lbs	2813 lbs

* Load applied 2ft from pole top

The result is a manufactured pole that has an actual load carrying capability that is 37.5% less than a wood pole. Manufacturers sometimes refer to these poles as "Grade B – Wood Pole Equivalent" because after Grade B transverse wind overload factors are applied, the poles will have equivalent load carrying capabilities.

Example:

Class 5 Wood Pole Grade B Transverse Wind Only - 1900 lbs / 4.0 = 475 lbs
Class 5 Metal Pole Grade B Transverse Wind Only – 1187 lbs / 2.5 = 475 lbs

Manufactured poles that are designed based on reduced tip loads cannot be modeled correctly using PoleForeman because the calculations in the program are based on wood pole dimensions and strengths as specified in ANSI 05.1. To model a manufactured pole using PoleForeman, the manufactured pole must have the same strength as a wood pole per ANSI 05.1.

It is recommended that you consult with your standards engineer or pole manufacturer to determine if the concrete and/or metal poles purchased by your company have the same load carrying capacity as wood poles (prior to application of NESC safety factors).

Modeling Stub Guy Poles

Modeling stub guy poles is a two-part process. First, the tension on the span guys must be determined. Next, the span guy tensions must be applied to the stub pole to determine if the stub pole has been properly sized and guyed. The following steps will guide you through the process.

Step 1 – Model Supporting Structure

Model the structure that will be supported by the stub guy pole (span guys). Once the structure has been modeled, view the pole in the Solid Model to verify the correctness of the attachments. Analyze the structure to determine proper pole sizing and guying. If the structure is properly sized and guyed then proceed to step 2.

Step 2 – Admin Login

Click *Tools* then *Admin.* in the menu bar. Enter your user name and "PoleForeman" as the password. Click *OK.*

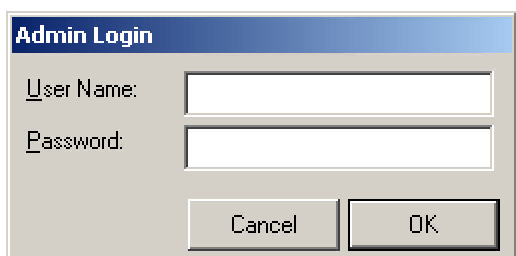


Figure A-1 Admin Login Screen

Step 3 – Load Factors

Select *Tools* and NESC from the menu bar. Click *Guys* then *Custom.* This sets all the load factors to unity (1). Click *OK.*

Step 4 – Calculate Span Guy Tension with No Load Factors

Select *Analysis*, then *Ice and Wind (250B)* from the menu bar. Record the tension for each span guy. The results displayed are without load factors applied.

Step 5 – Model Span Guys Using Joint Use Cables

Click *File*, then *New*. Click *No* when prompted to save file. To analyze the stub pole, joint use cables are used to model the span guys attached to the stub pole. This allows you to manually enter the span guy tensions calculated in step 4. Click *Install*, then *Span* in the menu bar. Enter the correct span length and direction. Select the span, click the *Joint Use* tab on the right side of the screen. For the number of Joint Use Cables, select the number of span guy strands.

Step 6 – Enter Span Guy Tensions

The joint use cable screen is shown in Figure A-2.

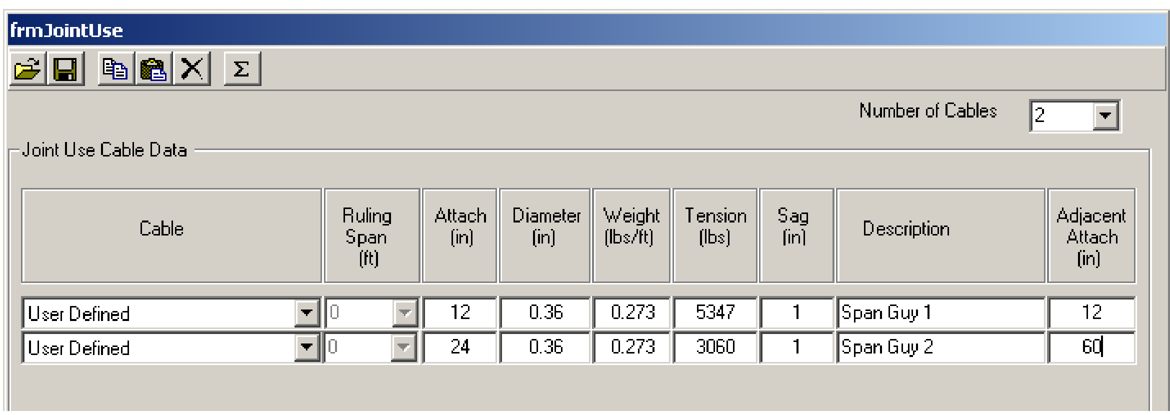


Figure A-2 – Joint Use Cable Screen

Select *User Defined* from the cable combo boxes. Enter the diameter, lbs/ft., tensions (from step 4), sag (assume 1"), and attachments for each span guy. The attachments are where the span guys are attached on the stub pole. Click OK when finished.

Step 7 – Add Anchors

If there are any anchors supporting the stub guy pole, add the anchors to the structure and enter associated data.

Step 8 – View Solid Model

View the Solid Model to verify the correctness of attachments.

Step 9 – Analyze Structure

Click *Analysis* and then *Ice and Wind (250B).* The program calculates the loading on the stub pole and displays the results.

Note

When the File New command was executed in step 5, the NESC overload factors were reset to the default values. The reason for setting the factors to unity in step 3 was to prevent the overload factors from being applied twice since this is a two-step process.

Conductor Uplift

The vertical force from conductor weight that a pole supports (Load Span) is the weight of the conductor between the low points in sag of the adjacent spans. On steep inclined spans, the low point in sag can theoretically fall beyond the lower structure. If this occurs, it indicates the conductor in the uphill span exerts uplift on the lower structure. The amount of uplift is equal to the weight of the conductor from the lower structure to the projected low point in sag. When the uplift of the span on one side of the structure is greater than the downward load of the span on the other side of the structure, then the structure is subject to an uplift force at the conductor support. In other words, the structure would be holding down the conductor rather than supporting it.

When the calculated Load Span is negative, as described above, conductor uplift is subject to occur. Uplift at a structure may cause the conductor to pull a suspension insulator up. With pin type insulators, uplift may cause the conductor to pull away from the insulator and possibly damage the insulator, pin, or arm.

The PoleForeman software will check for negative load spans. If a negative load span is calculated, the program will place a message on the *Analysis Screen* that states; "Pole is Subject to Conductor Uplift".

Spacer Cable

Spacer cable can be modeled in PoleForeman using methods similar to those used when modeling standard overhead construction. A typical three phase tangent spacer cable installation is shown in Figure A-3.

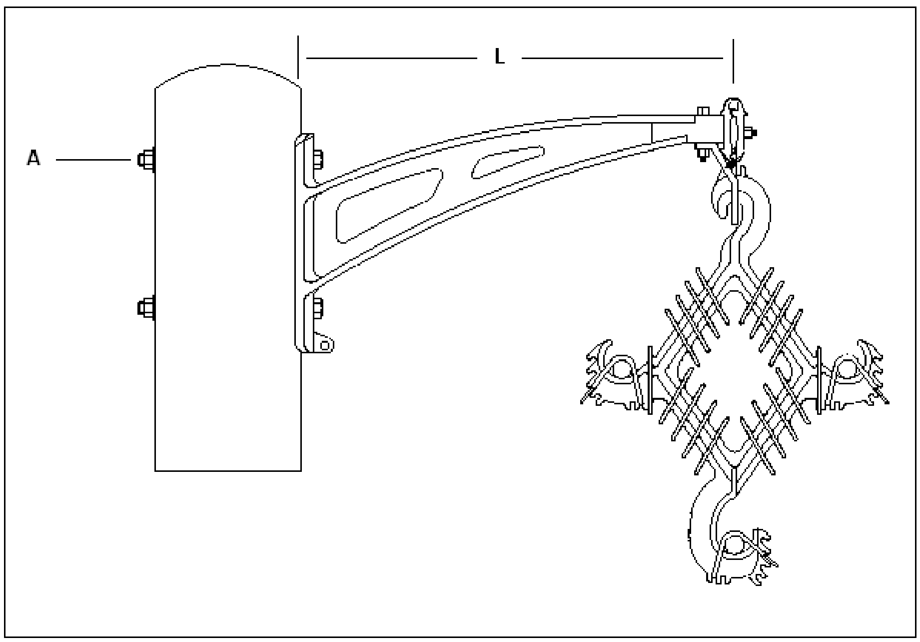


Figure A-3 Typical Tangent Spacer Cable Installation

The phase conductors for the spacer cable are specified in PoleForeman on the Power Conductor data tab for each span. The ruling span for the spacer cable phase conductors is assumed to be the distance between spacers. That is, if the spacing of the spacers is 30 ft., then enter 30 for the primary conductor ruling span.

The messenger for the spacer cable is specified on the Primary / Neutral data tab. It is important that the correct messenger be selected based upon the phase conductors that are to be installed. Lists of the available messengers are included in the Neutral conductor combo box. The ruling span for the messenger is calculated between dead-ends where as the ruling span for the phase conductors is assumed to be the distance between spacers.

Upon selecting the phase conductors and messenger for the spacer cable assembly, click the Construction Details command button to specify the brackets and insulators supporting the spacer cable. It is recommended that spacer cable framing units be utilized in lieu of manually specifying the brackets, insulators, attachments and offsets.

Estimating Joint Use Cable Tension

The Joint Use Data screen contains a tool that helps estimate the sag and tension associated with joint use cables that are supported by a steel messenger strand. The sag and tension calculations are based on field measurements that are used to derive the NESC loaded tension for the cable.

It is recommended that you consult with the owner of the joint use cable of interest to obtain the correct cable diameters, weights, sags and tensions. In the absence of said data, the following steps help estimate the sag and tension of a joint use cable supported by a steel messenger.

Step 1 – Select Span

On the Topology screen, click on the span in which the joint use cable is to be installed. The span is highlighted in red. Adjust the span length as necessary to reflect the actual field conditions or design conditions. Select the *Joint Use* data tab on the right side of the screen. Enter the number of joint use cables to be installed within the span. Click the *Cable Data* command button. The Joint Use Cable Data screen is displayed.

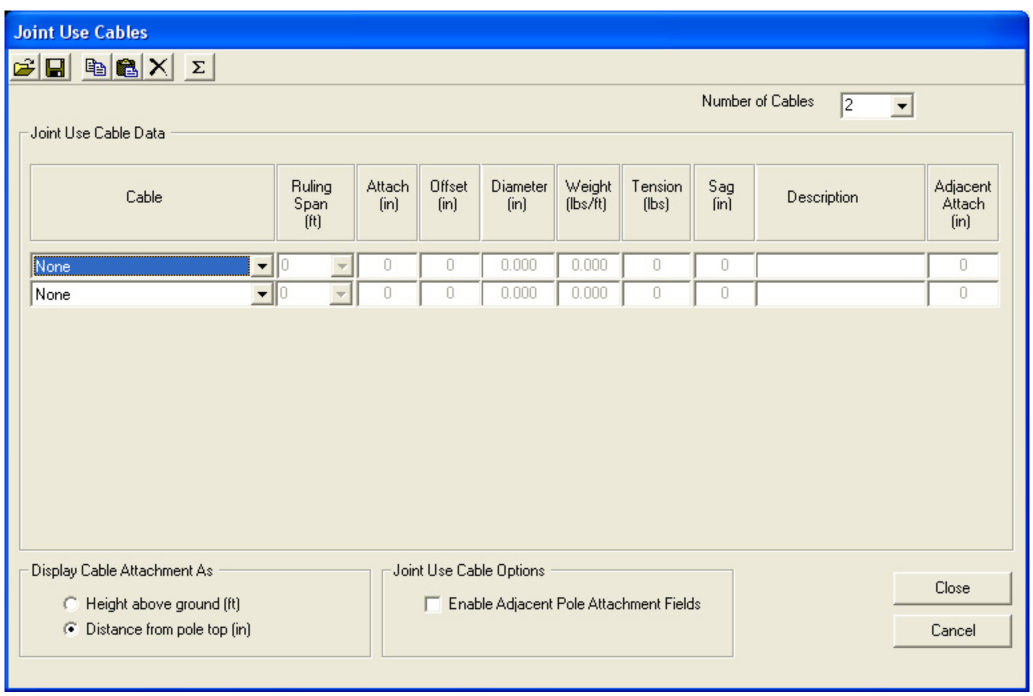


Figure A4 – Joint Use Cable Data Screen

Click one of the rows in the cable data table to make that row the active row. In the toolbar, click the Sag and Tension Σ button. The Cable Sag and Tension Calculator screen is displayed.

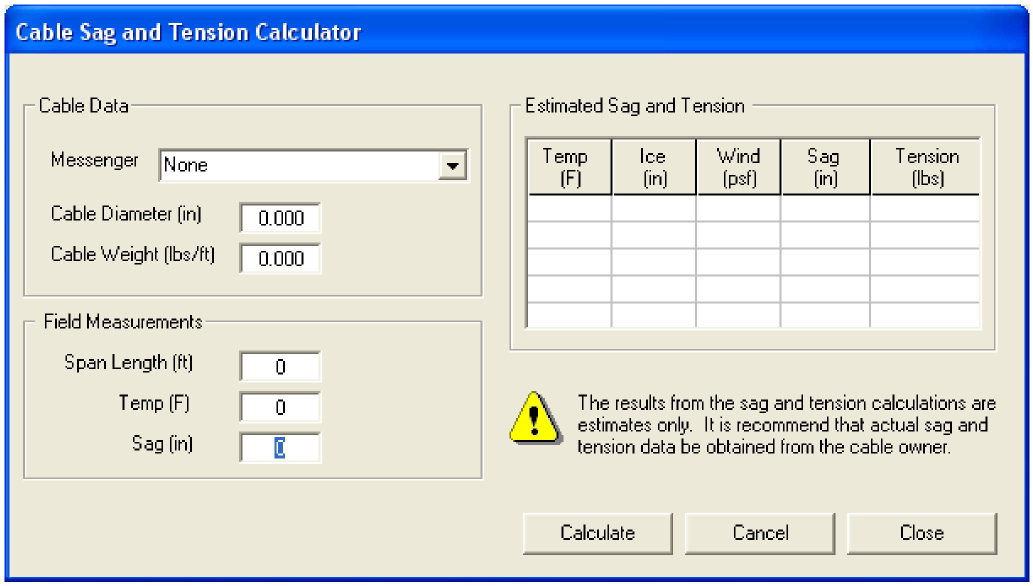


Figure A5 – Cable Sag and Tension Calculator

Step 2 – Messenger

In order to estimate the sag and tension of a joint use cable, the messenger strand size and type must be specified. Select a messenger strand from the messenger combo box.

Step 3 – Cable Diameter

Enter the overall diameter of the joint use cables and messenger bundle. The diameter is measured in inches.

Step 4 – Cable Weight

Enter the cable weight. The weight is measured in lbs/ft.

Step 5 – Span Length

Enter the span length for the cable. The span length is measured in feet.

Step 6 – Cable Temperature

Enter the temperature (F) of the cable at the time sag measurement is taken.

Step 7 – Enter Sag

Enter the sag measurement of the cable. For existing joint use cables, the sag can be obtained from field measurements. For new construction, consult with the owner of the cable. Some common assumptions are:

1. Assume the sag of the joint use cable will be installed at the same sag as the neutral supply conductor or other joint use cables in the span.

2. Assume the sag of the cable at 60° F to be 1.5% to 2.0% of the span length.

Note

There is no definitive data to support these assumptions. It is recommended that you consult with those in your local area that are responsible for installing aerial communication cables to establish appropriate assumptions for cable sag.

After entering the sag and temperature for the cable, click *Calculate*. The program attempts to estimate the tension of the cable under NESC loaded conditions. The results are displayed in the cable sag and tension table.

Note

The sag and tension data calculated are estimates only and do not take into consideration items such as permanent elongation and unequal support elevations. The calculations assume the steel messenger strand is the only component of the joint use cable assembly that is stressed.

Step 8 – Close

Click Close to unload the Cable Sag and Tension calculator. The data for the joint use cable is loaded into the joint use data table.

Foundation Stability of Direct-Embedded Poles

The ability of earth foundations to support direct embedded utility poles is dependent upon the forces acting on the pole, the pole setting depth and the physical properties of the soil. Common practice in the utility industry for distribution poles is to assume that standard pole setting depths of 10% of the pole length plus 2 feet will provide a stable foundation. Historically, the 10% plus 2ft rule-of-thumb has proven to be sufficient for typical distribution poles in good soil.

To properly design an earth foundation requires soil borings at the proposed pole locations to determine the strength properties of the soil. Since obtaining soil borings is not common practice for distribution poles, detailed foundation design is beyond the scope and capabilities of the PoleForeman software.

For companies that consider the simplified REA method of checking foundation stability and preliminary pole setting depths to be sufficient, the PoleForeman software does have the ability to perform these calculations. Refer to RUS Bulletin 1724E-200 dated August 2009 for additional information on the REA method. The REA method provides a simplified procedure of evaluating pole setting depths based on three soil classifications. The soil classifications are shown in Table A-2

Table A-2 Soil Classifications

Soil	Description	Soil Constant
Good	Very Dense, well graded sand and gravel, hard clay, dense, well graded, fine and coarse sand.	140
Average	Firm Clay, firm sand and gravel, compact sandy loom.	70
Poor	Soft Clay, poorly compacted sands (loose, coarse, or fine sand), wet clays and soft clayey silt.	35

Note

If no soil classification is selected for the analysis pole in the software, the program assumes a rigid foundation and does not check the foundation stability or pole setting depth.