Proposed workflow for PLS-POLE

A high-level sample of a proposed workflow for a typical project in PLS-POLE

Proposed workflow for PLS-POLE

The intent of this flow-chart is to assist both new and experienced users of our software with a handy and general reference for the typical decisions and steps needed to develop and model a complete PLS-POLE project.

The steps and processes are not exhaustive, and some projects may well require a different sequence to that proposed in this document.

Not all steps are mandatory/required for every type of project.

The main process flow can be comfortably printed/plotted on A0 size paper (47x33"). Although it is expected that this will remain as an electronic PDF file in most cases.

KEY:

Process

Process – with subsequent steps — *- linked to the Subprocess details*

Process – with subsequent steps.
Not necessary on every project

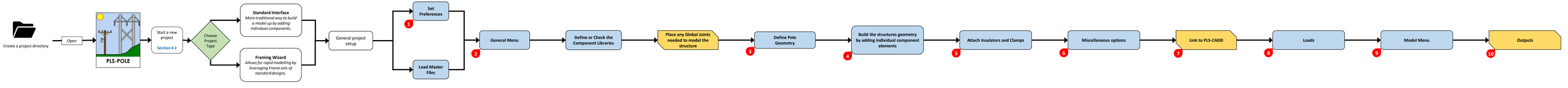

- Back to Main Process workflow


- Webinar Available for download through latest version of PLS-CADD


- YouTube Feature Overview video available


- Technical Note Available at https://www.powerlinesystems.com/technical-notes


- Sample files available on the PLS online Library


Create a project directory
Open
PLS-POLE
Start a new project
Section 4.1
Choose Project Type
Standard Interface
More traditional way to build a model up by adding individual components.
Framing Wizard
Allows for rapid modelling by leveraging Frame sets of standard designs.
General project setup
Set Preferences
Load Master Files
1
2
General Menu
Define or Check the Component Libraries
Place any Global Joints needed to model the structure
3
Define Pole Geometry
4
Build the structures geometry by adding individual component elements
5
Attach Insulators and Clamps
6
Miscellaneous options
7
Link to PLS-CADD
8
Loads
9
Model Menu
10
Outputs

Wood pole Buckling Assumptions
PLS-POLE can typically account for buckling by performing a non-linear analysis., however, there may be scenarios where other techniques to check buckling might be needed.

In these cases the options to control this are under:
General/ Wood pole Buckling Assumptions

Section 3.1.2.3.2 and 3.1.2.3.3

Post Processor Options
PLS-POLE users are provided with the ability to post-process the program results.
The XML options are the best way to extract data from a PLS-POLE run and the only ones supported for new applications.

General/ Post Processor Options
Section 4.2.3

Output Options
This menu gives the User control over what is produced after a Model is Run.

- *Controls the creation of and the contents within the Analysis Results and Summary Results Reports. Also allows for the Printing of Extended diagnostic output to allow further investigation into background calculations performed by the software.*
- *Allows for the creation of separate windows for:*
  - *Non-linear convergence graph*
  - *Usage summary graph*
  - *Base plate design*

You can also create customised Member Labels, and choose a Warning to print in any reports.

General/ Output Options
Section 4.2.2

General Data
Controls most of the information about the type of analysis and the project.

- *Analysis Type (Linear or Non-linear).*
- *Strength check type for Wood, Steel or FRP Poles.*
- *What type of Load (i.e. Transmission or Telecommunication)?*

As well as a few other important aspects that can have a bearing on the model.

General/ General Data
Section 4.2.1

General Menu:
The General Menu has many important settings

Define Pole Geometry
Section 4.4

Select Pole Material

- **Wood:** *Naturally grown*
- **Steel:** *Tubular Steel*
- **Concrete**
- **Laminated Wood**
- **FRP:** *Fibre Reinforced Polymer ~ Composite*
- **Masts:** *Modular Latticed Masts ~Emergency Response Structures*

Number of Poles in Model?

Place a Single Pole

Place Multiple Poles

*Use the **Add/Move/Delete Toolbar** button:*
Add pole
Add a pole to the model

By the X, Y and Z of base and X, Y inclination angles.
This should be used for single poles and simple frames.
Geometry/ Pole Material...

By tip and base joint.
This is only appropriate for A-Frame, Y-Frame and other complicated structure's
Geometry/ Pole Material...

Wood Pole Specific options:
The following options are only available when modelling Wood Poles.

Push Brace:
The Push Brace dialog box/tool allows you to quickly and easily model the selected wood pole as a push brace with a pinned connection if needed.

Top & Bottom Cut Length:
You can cut a fixed length at the top of the pole, thus decreasing its total length and increasing its top diameter. You can also cut a fixed length at the bottom of the pole.

*Select the **Property** and the **Material (or Wood Pole Species)** from the respective Component Libraries based on the pole material at the first decision point.*

Define Attachment Labels
These can be defined either by the Distance from the Tip downwards, or by a Global Z coordinate. This allows you to manually connect other Elements or Insulators to the Pole/s at the specified positions.

Define Base Connection type
This can either be:

- ***Fixed**: Unable to move or rotate.*
- ***Pinned**: cannot move, but can rotate about the X and Y axes (torsional restraint is provided).*
- ***PinFrm**: cannot move, but CAN rotate about all axes.*

*If you have placed your poles by joints, the connectivity can rather be defined under **Geometry/ Joints**.*

Embedment Depth Override:
Can override the Default Embedded Depth captured in the Component Library.

The overriding embedment length is made up of two parts. A fixed part, the "Embedment Constant", plus a variable part which is a percentage, "Embedment %", of the total length of the pole.
Therefore, the overriding embedment length is equal to "Embedment Constant" + "Embedment %" x Total Pole Length. The "Embedment %" field is entered as a percent.

Multiple Pole Selection
You can select one or more poles for which you wish to determine the allowable spans or interaction diagrams between allowable spans.

Optimization
PLS-POLE has a few built in optimization routines. These tools offer an Engineer the opportunity to attempt multiple combinations of pole, arm or cross-brace properties and locations, in an attempt to find the lightest possible elements.
This can be far faster and more accurate than a simple trial and error approach which is the alternative.

The optimization routines that are available are:

- **Model/ Optimum Pole Selection**
- **Model/ Steel Pole Shaft Optimizer**
- **Model/ X-Brace Optimizer**
- **Model/ Tubular Steel Davit Arm Optimizer**

Section 6.1, 6.2, 6.3 and 6.4

Deformed Geometry View
This view provides you with a graphical summary of the analysis results. The picture that you see depends on the selections made in the 3-d Controls dialog box. That 3-d Controls dialog box is opened by clicking on the 3D button.

View/ Display Options/ Set Rotation, Color and Label Options
Section 2.5

Run
*Once the model has been checked, its analysis can be performed with the **Model/ Run** command, as long as you have allocated or defined a loading file (.lca, .lic or .eia)*

Once the analysis is complete, typically the following windows are opened:

- *the **Deformed Geometry***
- *a long text report (labeled **Analysis Results**) and*
- *a short report (labeled **Summary**)*

*You can see all windows simultaneously with **Window/ Tile Horizontal** or **Window/ Tile Vertical**.*

*The menu command **File/Analyze Multiple Models...**, will allow for the selection and analysis of multiple structure files. This feature opens models in read-only mode to avoid errors if files can't be locked.*

Section 2.3

Results reports
The reports can be viewed in PLS-POLE, or can be saved as Text or PDF files.
Additionally, if you right-click in any result table you can enter a table view mode. This mode allows for the sorting and filtering of the results/data, as well as extraction copying for use in other programs.

Additionally, all tables and reports can be exported as an XML file for outside post-processing.

Section 2.4

Check
*Once your model is complete and loads have been assigned, it is good practice to initially check the model to see if there are any **Warnings** or **Errors***

*The program is dynamic in that it will flag you during the modelling process if there are any **Errors** or **Warnings** with text and visual prompts, however the **Model/ Check** is a concise list for the whole model.*

Linking to PLS-CADD parts and assemblies management system
If you make use of material tracking in the PLS System and have an existing Part/ Assembly Library (.prt file) with the corresponding stock numbers being used for the structural components in the structural program, then PLS-CADD can automate material take-off and costing for and entire line project.*

Geometry/ PLS-CADD/ ...
Appendix B.2

Mapping Attachment Points to PLS-CADD Cable Sets
If you wish to use your structure models into PLS-CADD you need to provide a link or map between the load points on a structure (the tips of insulators and clamps) and the Conductor Sets for use in PLS-CADD.

Geometry/ PLS-CADD/ Insulator Link
Appendix B.1

Imperial (U.S.)
Metric (S.I.)
What Unit System should be used?

File and Directory Mapping
*The location of default locations for the directories or library files is mapped here. Normally only need to define the **Structure Directory**, the **Cable and Marker Balls Directory** and the **Part/Assembly Library** (.prt file).*
The Schema file Schema file (.sma) for personal customizations such as report or toolbars is also mapped and selected here.

*Apply a variety of **Settings** or **Mini-Preferences.** These control a range of attributes or settings from unit preferences, display settings and many more.*

*Select the **Fonts and Colors** for various reports and graphics views generated by the program and control the **View background color.***

Set Preferences
*Preferences are set and controlled under the **File/Preferences** menu.*
*When you **OK** the Preferences dialog, all the preference settings, are automatically saved in a file named PLS_CADD.**ini** which is located in your WINDOWS directory or in "C:\Users\<user name>\Appdata\Roaming\PLS\pls_cadd.ini".*
The file name may vary based on the WINDOWS installation. These preferences remain in effect until changed.

Section 2.1.1

Component Libraries:
Component Libraries are defined at length in Section 3 of the Manual.
In many cases the data needed or these various libraries should be defined by the User. However, there is a large range of available libraries that are available on the Power Line Systems website for download, but these can be accessed directly through the Software.

Section 2.1.1.1

Create a project directory
Open
PLS-POLE
Start a new project
Section 4.1
Choose Project Type

Standard Interface
More traditional way to build a model up by adding individual components.

Framing Wizard
Allows for rapid modelling by leveraging Frame sets of standard designs.

General project setup

1 **Set Preferences**
Load Master Files

2 **General Menu**

Define or Check the Component Libraries

Place any Global Joints needed to model the structure

3 **Define Pole Geometry**

4 **Build the structures geometry by adding individual component elements**

5 **Attach Insulators and Clamps**

6 **Miscellaneous options**

7 **Link to PLS-CADD**

8 **Loads**

9 **Model Menu**

10 **Outputs**

Loads
Design loads on transmission poles/ frames include wire loads (from conductors and ground wires), dead loads from the structure and permanently attached components, wind loads on the structure itself and possibly ice loads on the members.

Depending on whether PLS-POLE is run for transmission/ distribution structures in design check mode, or in one of the allowable span modes (Sections 1.1.2 and 4.2), or is run for communication structures in the special EIA design check mode, the design loads have to be specified in standard load files with the vector loads format (".lca"), the wire loads format (".lic"), or the EIA loads format (".eia").

Sections 5.3 to 5.6.

Batch Import or Edit Concentrated Loads
This command will reference a table version of the vector load file for all attachment points. This table can be edited or exported with options that are accessed when right-clicking within the table. Any edits will be saved with the vector file (.lca) that is referenced with the structure.

This can be very useful if importing loads from an external calculator/Excel file.

Sections 5.3.

Load trees
*A summary table of the loading on the Pole structure can be obtained by selecting **Loads/ Loading Tree Report***

A more graphical Load tree can be obtained as using the Drafting tools.
Section 8.5

Pole drawings
Scaled drawing sheets can be produced automatically by PLS-POLE according to user specified parameters. You have complete flexibility in choosing scales and page appearance.
The drawings can be sent directly to a Windows® printer, or they can be exported as DXF files readable by most CAD systems (AutoCAD ®, Microstation ®, etc.) for further enhancements, or they (together with other project documents) can automatically be converted to PDF documents as a truly universal way to share them with all interested parties.

Drawings can be embellished further by adding Inset Views and even attaching external files to be used.

Section 8

Build the structures geometry by adding individual component elements
*Use the **Add/Move/Delete Toolbar** button*
Add element
Add an element
*Or use the tools in the **Geometry/** menu*

Add Brace:
Braces are prismatic components with uniform cross section properties throughout their length.
*A brace is modeled internally as a single straight bar with either unlimited axial capacity (**standard** or truss element) or limited tension and compression capacities (**fuse** element).*
Can be added between any pair of joints with predefined labels.

Section 4.7.1

Add Guy:
Guy wires are attached to structures to resist loads and prevent large deflections. Guys always connect to a fixed anchor joint

Section 4.7.2

Add Cables:
Cables are similar to Guys, but they must be connected between 2 joints. These can be used to model Span guys, tension only slender cable elements.

Section 4.7.3

Add Davit Arm:
***Davit arms** are prismatic components with a uniform cross-section along its length.*
*A **Tubular steel davit arm** is made out of steel and the cross section shape can be any one of the shapes available for tubular poles, it **can** be tapered along its length.*

Generic or Tubular Davit Arms can be attached rigidly to any joint with a pre-defined label. Davit arms can be oriented in any direction around the pole, by specifying the azimuth angle.
You can add intermediate joints along the length of the member to change the shape, make it inclined or represent a curve.

***Geometry/ Davit Arms** or **Geometry/ Tubular Steel Davit Arms**.*
Section 4.5

Add Cross Arm:
***Cross arms** are either straight prismatic or steel angle components. **Tubular steel cross arms** are made out of steel and the cross section shape can be any one of the shapes available for tubular poles, it **cannot** be tapered.*

Generic or Tubular Cross Arms can be connected to a single joint or to any number of joints with predefined labels.

***Geometry/ Cross Arms** or **Geometry/ Tubular Steel Cross Arms**.*
Section 4.5

Crossarm elements can now be single angle members.
Section 3.3.1

Add Connection and Anchor (CAN) Element:
A CAN element can be used to check the capacity of joints due to forces or moments passed from other elements such as braces, cables, guys, crossarms, davit arms, poles, etc. A CAN element can be attached to any joint in the model and you may select up to four elements to pass forces and moments to the CAN.

A CAN element has no graphical display and no physical attributes.
Section 4.14

Add Framing:
A Framing is a collection/grouping of structural elements that do not include the pole itself. These can be added onto an existing pole model to rapidly create a model of a standard configuration.

*The **Components/ Framing/ Manager** is used to not only place the framing graphically on a model as discussed in Section 4.16 but also to edit the framing in case you need to change something in the framing library.*

Add Vang
(Not in a library):
Vangs are small components used to attach insulators or other items to the face of a pole, crossarm or davit arm. They are not modelled as structural elements and act as a load transfer mechanism.

Geometry/ Vangs
Section 4.4.7

Add Loads from Permanently attached Equipment
(Not in A Library):
Design loads from attached equipment that is not actually modelled can be placed at specified joints by:

Geometry/ Miscellaneous/ Dead Loads and Drag Areas
Section 4.9

Add Equipment (from a Library):
Equipment can be added to a pole model to represent the various pieces of equipment placed on poles. These can include:

- *Switches*
- *Transformers*
- *Capacitor banks*
- *Voltage regulators*
- *Metering cabinets*
- *Telecommunication antennae, etc...*

Equipment is placed at a joint and then located by adding azimuth angle and an offset distance from the face of the pole.
Geometry/ Equipment
Section 4.8

Attach Insulators and Clamps:
Insulators and attachment devices (clamps) should always be used where you intend to transfer ground wire or conductor loads to the structure. You cannot apply such loads directly to the structure. They have to be applied to tip labels on clamps or insulators.

*Use the **Add/Move/Delete Toolbar** button*
Add insulator
Add an insulator

*Or use the tools in the **Geometry/ Insulators/***

Section 4.10

Add Clamp:
A clamp is a device that transfers concentrated loads from ground wires, conductors or other sources to an attachment point on the structure. A clamp has no physical dimension and no weight, but has strength properties.

Add Strain:
A strain insulator is normally used to dead-end a ground wire or conductor to a structure. It is not modeled as a structural element, but as a load transfer mechanism.

Add Suspension:
A suspension insulator normally supports a ground wire or a conductor. It is not modeled as a structural element, but as a load transfer mechanism.

Add 2-Part:
A 2-parts insulator normally supports a conductor at the junction point of two insulators. The most common 2-parts insulators are the V-strings

Add Post:
A post insulator is modeled as a short cantilevered beam element rigidly attached at its origin to the structure model. Posts can be horizontal, vertical or upswept (inclined).
Posts can optionally include brace elements.

Add Guy Strain:
Guy strain insulators are applied to down guys in order to provide insulation to down guys from energized portions of the other insulators or cables in PLS-CADD.

Find Allowable Swing Angles
PLS-POLE is capable of calculating the allowable swings of suspension insulators from the size of the energized zones associated with the insulators and attached hardware and the required electrical clearances to the various components of the pole including guy strain insulators.

Geometry/ Insulators/ Find Allowable Suspension Swing Angles

Section 4.10.1

Calculating Allowable Load Angles for 2-part Insulators
PLS-POLE can determine the allowable load angles for 2-part insulators. The allowable load angles will guarantee that neither side of the 2-part insulator will ever go into compression.

Geometry/ Insulators/ Calculate Allowable 2-Part Load Angles

Section 4.10.2

Foundation Properties:
You can enter both the Strength and Stiffness data for the foundations of Pole models.

So, the actual foundation is not modelled, but the capacity of the foundation system can be checked by PLS-POLE during the analysis.

Geometry/Miscellaneous/Foundation Strength....

Section 4.11

Caisson Foundation Design and Analysis:
If you make use of direct embedded and drilled concrete pier foundations PLS-POLE can analyse and design this type of system using the CAISSON program (as long as this has been purchased in addition to the PLS-POLE program)

To make use of this functionality you must:

- *Enter the physical characteristics of the foundation in the **Geometry/ Miscellaneous/ Foundation Strength**, and*
- *The soil and CAISSON settings shall be entered in **Geometry/ Miscellaneous/ Soil and Caisson Settings***

Section 4.11.3

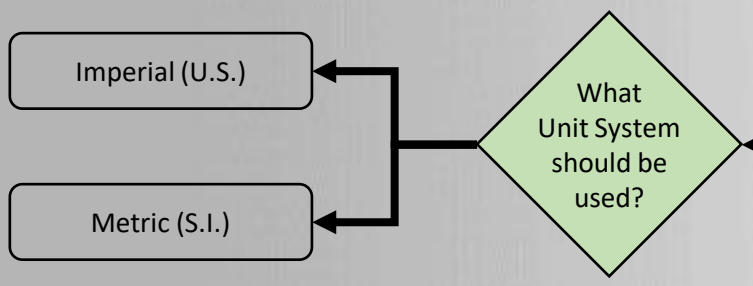
Wood Pole Defects:
Several defects that can reduce the strength of wooden utility poles can be modelled within PLS-POLE. These could be used to simulate drilling holes, woodpecker damage, rot/decay or others.

The types of defect that an be defined are:

- *Enclosed pocket*
- *External Decay*
- *External Pocket*
- *Hollow Heart*
- *Thru-hole*

*These are captured under the **Geometry/ Miscellaneous/ Wood Pole Defects** menu.*

Section 3.9 and Section 4.4.2.1



File and Directory Mapping

The location of default locations for the directories or library files is mapped here. Normally only need to define the ***Structure Directory****, the* ***Cable and Marker Balls Directory*** *and the* ***Part/Assembly Library*** *(.prt file).*
*The Schema file Schema file (****.sma****) for personal customizations such as report or toolbars is also mapped and selected here.*

Apply a variety of **Settings** *or* **Mini-Preferences.**
These control a range of attributes or settings from unit preferences, display settings and many more.

Select the **Fonts and Colors** *for various reports and graphics views generated by the program and control the* **View background color.**

Set Preferences

Preferences are set and controlled under the **File/Preferences** *menu.*
When you **OK** *the Preferences dialog, all the preference settings, are automatically saved in a file named PLS_CADD.****ini*** *which is located in your WINDOWS directory or in "C:\Users\<user name>\Appdata\Roaming\PLS\pls_cadd.ini".*
The file name may vary based on the WINDOWS installation.
These preferences remain in effect until changed.

Section 2.1.1

1

General Menu:
The General Menu has many important settings

2

General Data
Controls most of the information about the type of analysis and the project.

- *Analysis Type (Linear or Non-linear).*
- *Strength check type for Wood, Steel or FRP Poles.*
- *What type of Load (i.e. Transmission or Telecommunication)?*

As well as a few other important aspects that can have a bearing on the model.

General/ General Data
Section 4.2.1

Output Options
This menu gives the User control over what is produced after a Model is Run.

- *Controls the creation of and the contents within the Analysis Results and Summary Results Reports. Also allows for the Printing of Extended diagnostic output to allow further investigation into background calculations performed by the software.*
- *Allows for the creation of separate windows for:*
  - *Non-linear convergence graph*
  - *Usage summary graph*
  - *Base plate design*

You can also create customised Member Labels, and choose a Warning to print in any reports.

General/ Output Options
Section 4.2.2

Post Processor Options
PLS-POLE users are provided with the ability to post-process the program results.
The XML options are the best way to extract data from a PLS-POLE run and the only ones supported for new applications.

General/ Post Processor Options
Section 4.2.3

Wood pole Buckling Assumptions
PLS-POLE can typically account for buckling by performing a non-linear analysis., however, there may be scenarios where other techniques to check buckling might be needed.

In these cases the options to control this are under:
General/ Wood pole Buckling Assumptions

Section 3.1.2.3.2 and 3.1.2.3.3

Define Pole Geometry

Section 4.4

3

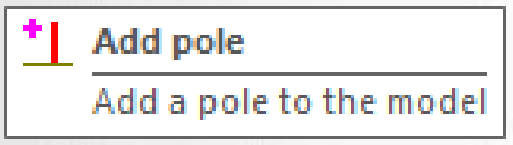
Select Pole Material

- **Wood:** *Naturally grown*
- **Steel:** *Tubular Steel*
- **Concrete**
- **Laminated Wood**
- **FRP:** *Fibre Reinforced Polymer ~ Composite*
- **Masts:** *Modular Latticed Masts ~Emergency Response Structures*

Number of Poles in Model?

- **Place a Single Pole**
- **Place Multiple Poles**

*Use the **Add/Move/Delete** Toolbar button:*



By the X, Y and Z of base and X, Y inclination angles.

This should be used for single poles and simple frames.

Geometry/ Pole Material…

By tip and base joint.

This is only appropriate for A-Frame, Y-Frame and other complicated structure's

Geometry/ Pole Material…

*Select the **Property** and the **Material (or Wood Pole Species)** from the respective Component Libraries based on the pole material at the first decision point.*

Define Attachment Labels

These can be defined either by the Distance from the Tip downwards, or by a Global Z coordinate.
This allows you to manually connect other Elements or Insulators to the Pole/s at the specified positions.

Define Base Connection type

This can either be:

- ***Fixed**: Unable to move or rotate.*
- ***Pinned**: cannot move, but can rotate about the X and Y axes (torsional restraint is provided).*
- ***PinFrm**: cannot move, but CAN rotate about all axes.*

*If you have placed your poles by joints, the connectivity can rather be defined under **Geometry/ Joints**.*

Embedment Depth Override:

Can override the Default Embedded Depth captured in the Component Library.

The overriding embedment length is made up of two parts. A fixed part, the "Embedment Constant", plus a variable part which is a percentage, "Embedment %", of the total length of the pole.
Therefore, the overriding embedment length is equal to "Embedment Constant" + "Embedment %" x Total Pole Length. The "Embedment %" field is entered as a percent.

Multiple Pole Selection

You can select one or more poles for which you wish to determine the allowable spans or interaction diagrams between allowable spans.

Define Pole Geometry

Section 4.4

3

Select Pole Material

Wood: *Naturally grown*

Steel: *Tubular Steel*

Concrete

Laminated Wood

FRP:
Fibre Reinforced Polymer
~ Composite

Masts:
Modular Latticed Masts
~Emergency Response Structures

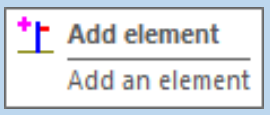
Wood Pole Specific options:
The following options are only available when modelling Wood Poles.

Push Brace:
The Push Brace dialog box/tool allows you to quickly and easily model the selected wood pole as a push brace with a pinned connection if needed.

Top & Bottom Cut Length:
You can cut a fixed length at the top of the pole, thus decreasing its total length and increasing its top diameter. You can also cut a fixed length at the bottom of the pole.

Build the structures geometry by adding individual component elements

Use the ***Add/Move/Delete*** **Toolbar** *button*



Or use the tools in the ***Geometry/*** *menu*

4

Add Brace:

Braces are prismatic components with uniform cross section properties throughout their length.
*A brace is modeled internally as a single straight bar with either unlimited axial capacity (**standard** or truss element) or limited tension and compression capacities (**fuse** element).*
Can be added between any pair of joints with predefined labels.

Section 4.7.1

Add Connection and Anchor (CAN) Element:

A CAN element can be used to check the capacity of joints due to forces or moments passed from other elements such as braces, cables, guys, crossarms, davit arms, poles, etc. A CAN element can be attached to any joint in the model and you may select up to four elements to pass forces and moments to the CAN.
A CAN element has no graphical display and no physical attributes.

Section 4.14

Add Guy:

Guy wires are attached to structures to resist loads and prevent large deflections. Guys always connect to a fixed anchor joint

Section 4.7.2

Add Framing:

A Framing is a collection/grouping of structural elements that do not include the pole itself. *These can be added onto an existing pole model to rapidly create a model of a standard configuration.*
The ***Components/ Framing/ Manager*** *is used to not only place the framing graphically on a model as discussed in* ***Section 4.16*** *but also to edit the framing in case you need to change something in the framing library.*

Add Cables:

Cables are similar to Guys, but they must be connected between 2 joints. These can be used to model Span guys, tension only slender cable elements.

Section 4.7.3

Add Vang (Not in a library):

Vangs are small components used to attach insulators or other items to the face of a pole, crossarm or davit arm. They are not modelled as structural elements and act as a load transfer mechanism.

Geometry/ Vangs

Section 4.4.7

Add Davit Arm:

Davit arms *are prismatic components with a uniform cross-section along its length.*
A ***Tubular steel davit arm*** *is made out of steel and the cross section shape can be any one of the shapes available for tubular poles, it* ***can*** *be tapered along its length.*
Generic or Tubular Davit Arms can be attached rigidly to any joint with a pre-defined label. Davit arms can be oriented in any direction around the pole, by specifying the azimuth angle.
You can add intermediate joints along the length of the member to change the shape, make it inclined or represent a curve.
Geometry/ Davit Arms *or* ***Geometry/ Tubular Steel Davit Arms****.*

Section 4.5

Add Loads from Permanently attached Equipment (Not in A Library):

Design loads from attached equipment that is not actually modelled can be placed at specified joints by:
Geometry/ Miscellaneous/ Dead Loads and Drag Areas

Section 4.9

Add Cross Arm:

Cross arms *are either straight prismatic or steel angle components.*
Tubular steel cross arms *are made out of steel and the cross section shape can be any one of the shapes available for tubular poles, it* ***cannot*** *be tapered.*
Generic or Tubular Cross Arms can be connected to a single joint or to any number of joints with predefined labels.
Geometry/ Cross Arms *or* ***Geometry/ Tubular Steel Cross Arms****.*

Section 4.5

Crossarm elements can now be single angle members.

Section 3.3.1

Add Equipment (from a Library):

Equipment can be added to a pole model to represent the various pieces of equipment placed on poles.
These can include:

- *Switches, Transformers, Capacitor banks,*
- *Voltage regulators, Metering cabinets,*
- *Telecommunication antennae, etc,..*

Equipment is placed at a joint and then located by adding azimuth angle and an offset distance from the face of the pole.

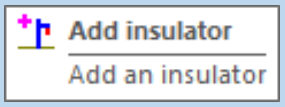
Geometry/ Equipment

Section 4.8

Attach Insulators and Clamps:

Insulators and attachment devices (clamps) should always be used where you intend to transfer ground wire or conductor loads to the structure. You cannot apply such loads directly to the structure. They have to be applied to tip labels on clamps or insulators.

Use the ***Add/Move/Delete*** **Toolbar** *button*



Or use the tools in the ***Geometry/ Insulators/***

Section 4.10

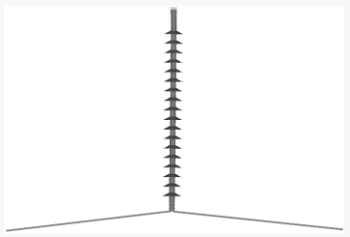
5

Add Clamp:

A clamp is a device that transfers concentrated loads from ground wires, conductors or other sources to an attachment point on the structure. A clamp has no physical dimension and no weight, but has strength properties.

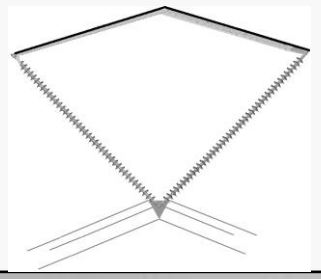
Add Strain:

A strain insulator is normally used to dead-end a ground wire or conductor to a structure. It is not modeled as a structural element, but as a load transfer mechanism.



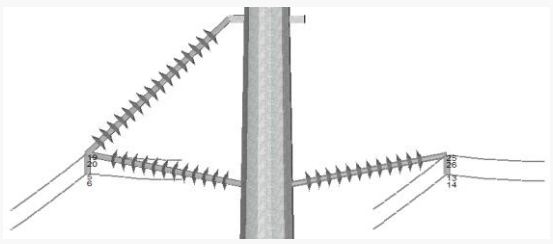
Add Suspension:

A suspension insulator normally supports a ground wire or a conductor. It is not modeled as a structural element, but as a load transfer mechanism.



Add 2-Part:

A 2-parts insulator normally supports a conductor at the junction point of two insulators. The most common 2-parts insulators are the V-strings



Add Post:

A post insulator is modeled as a short cantilevered beam element rigidly attached at its origin to the structure model.

Posts can be horizontal, vertical or upswept (inclined).

Posts can optionally include brace elements.

Add Guy Strain:

Guy strain insulators are applied to down guys in order to provide insulation to down guys from energized portions of the other insulators or cables in PLS-CADD.

Find Allowable Swing Angles

PLS-POLE is capable of calculating the allowable swings of suspension insulators from the size of the energized zones associated with the insulators and attached hardware and the required electrical clearances to the various components of the pole including guy strain insulators.

Geometry/ Insulators/ Find Allowable Suspension Swing Angles

Section 4.10.1

Calculating Allowable Load Angles for 2-part Insulators

PLS-POLE can determine the allowable load angles for 2-part insulators.

The allowable load angles will guarantee that neither side of the 2-part insulator will ever go into compression.

Geometry/ Insulators/ Calculate Allowable 2-Part Load Angles

Section 4.10.2

Miscellaneous options

6

Foundation Properties:

You can enter both the Strength and Stiffness data for the foundations of Pole models.
So, the actual foundation is not modelled, but the capacity of the foundation system can be checked by PLS-POLE during the analysis.

Geometry/Miscellaneous/Foundation Strength....

Section 4.11

Caisson Foundation Design and Analysis:

*If you make use of direct embedded and drilled concrete pier foundations PLS-POLE can analyse and design this type of system using the CAISSON program (*as long as this has been purchased in addition to the PLS-POLE program*)*

To make use of this functionality you must:

- *Enter the physical characteristics of the foundation in the* **Geometry/ Miscellaneous/ Foundation Strength***, and*
- *The soil and CAISSON settings shall be entered in* **Geometry/ Miscellaneous/ Soil and Caisson Settings**

Section 4.11.3

Wood Pole Defects:

Several defects that can reduce the strength of wooden utility poles can be modelled within PLS-POLE. These could be used to simulate drilling holes, woodpecker damage, rot/decay or others.

The types of defect that an be defined are:

- *Enclosed pocket*
- *External Decay*
- *External Pocket*
- *Hollow Heart*
- *Thru-hole*

These are captured under the **Geometry/ Miscellaneous/ Wood Pole Defects** *menu.*

Section 3.9 *and* ***Section 4.4.2.1***

Link to PLS-CADD

7

Linking to PLS-CADD parts and assemblies management system

If you make use of material tracking in the PLS System and have an existing Part/ Assembly Library (.prt file) with the corresponding stock numbers being used for the structural components in the structural program, then PLS-CADD can automate material take-off and costing for and entire line project.*

Geometry/ PLS-CADD/ …

Appendix B.2

Mapping Attachment Points to PLS-CADD Cable Sets

If you wish to use your structure models into PLS-CADD you need to provide a link or map between the load points on a structure (the tips of insulators and clamps) and the Conductor Sets for use in PLS-CADD.

Geometry/ PLS-CADD/ Insulator Link

Appendix B.1

Loads

8

Loads

Design loads on transmission poles/ frames include wire loads (from conductors and ground wires), dead loads from the structure and permanently attached components, wind loads on the structure itself and possibly ice loads on the members.

*Depending on whether PLS-POLE is run for transmission/ distribution structures in design check mode, or in one of the allowable span modes (**Sections 1.1.2 and 4.2**), or is run for communication structures in the special EIA design check mode, the design loads have to be specified in standard load files with the vector loads format ("**.lca**"), the wire loads format ("**.lic**"), or the EIA loads format ("**.eia**").*

***Sections 5.3 to 5.6**.*

Batch Import or Edit Concentrated Loads

This command will reference a table version of the vector load file for all attachment points. This table can be edited or exported with options that are accessed when right-clicking within the table. Any edits will be saved with the vector file (.lca) that is referenced with the structure.

This can be very useful if importing loads from an external calculator/Excel file.

Sections 5.3.

Model Menu

9

Check

*Once your model is complete and loads have been assigned, it is good practice to initially check the model to see if there are any **Warnings** or **Errors***

*The program is dynamic in that it will flag you during the modelling process if there are any **Errors** or **Warnings** with text and visual prompts, however the* **Model/ Check** *is a concise list for the whole model.*

Run

*Once the model has been checked, its analysis can be performed with the **Model/ Run** command, as long as you have allocated or defined a loading file (.lca, .lic or .eia)*

Once the analysis is complete, typically the following windows are opened:

- *the **Deformed Geometry***
- *a long text report (labeled **Analysis Results**) and*
- *a short report (labeled **Summary**)*

You can see all windows simultaneously with **Window/ Tile Horizontal** *or* **Window/ Tile Vertical**.

The menu command **File/Analyze Multiple Models...** *will allow for the selection and analysis of multiple structure files. This feature opens models in read-only mode to avoid errors if files can't be locked.*

Section 2.3

Optimization

PLS-POLE has a few built in optimization routines. These tools offer an Engineer the opportunity to attempt multiple combinations of pole, arm or cross-brace properties and locations, in an attempt to find the lightest possible elements.
This can be far faster and more accurate than a simple trial and error approach which is the alternative.

The optimization routines that are available are:

- **Model/ Optimum Pole Selection**
- **Model/ Steel Pole Shaft Optimizer**
- **Model/ X-Brace Optimizer**
- **Model/ Tubular Steel Davit Arm Optimizer**

Section 6.1, 6.2, 6.3 and 6.4

Deformed Geometry View

This view provides you with a graphical summary of the analysis results. The picture that you see depends on the selections made in the 3-d Controls dialog box. That 3-d Controls dialog box is opened by clicking on the 3D button.

View/ Display Options/ Set Rotation, Color and Label Options

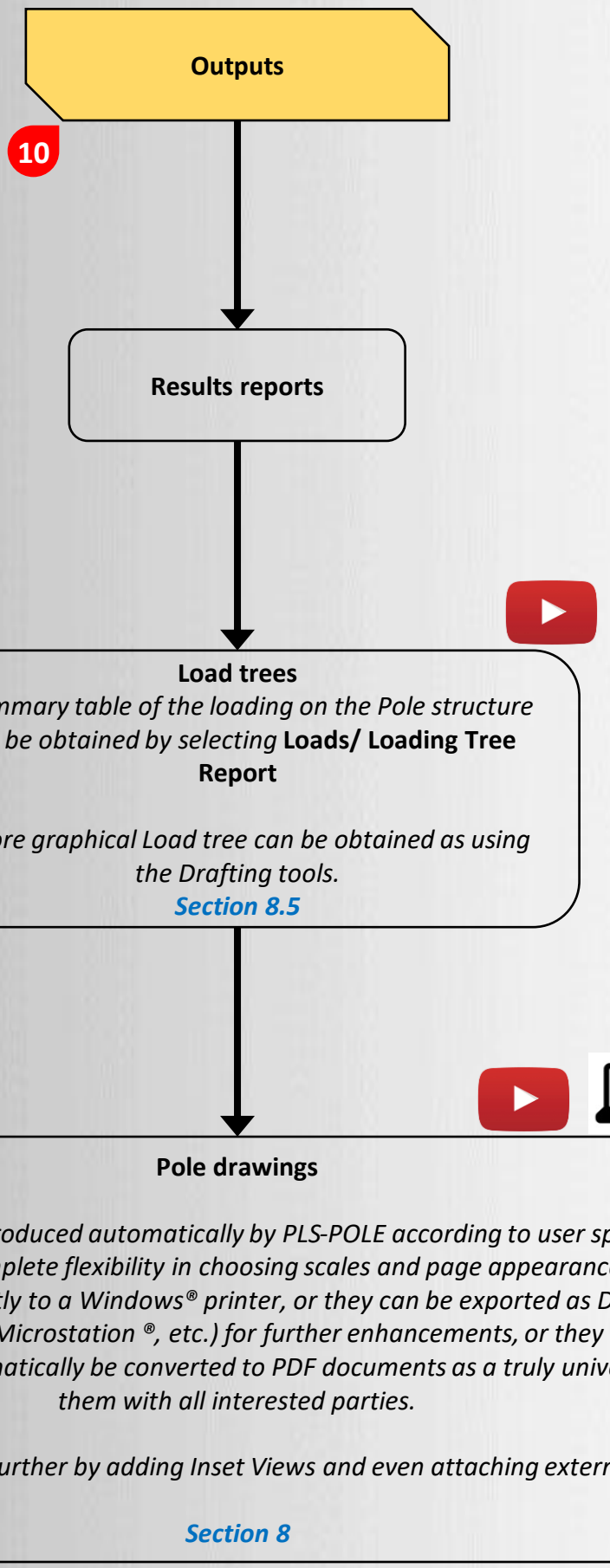
Section 2.5

Results reports

The reports can be viewed in PLS-POLE, or can be saved as Text or PDF files.
Additionally, if you right-click in any result table you can enter a table view mode. This mode allows for the sorting and filtering of the results/data, as well as extraction copying for use in other programs.

Additionally, all tables and reports can be exported as an XML file for outside post-processing.

Section 2.4

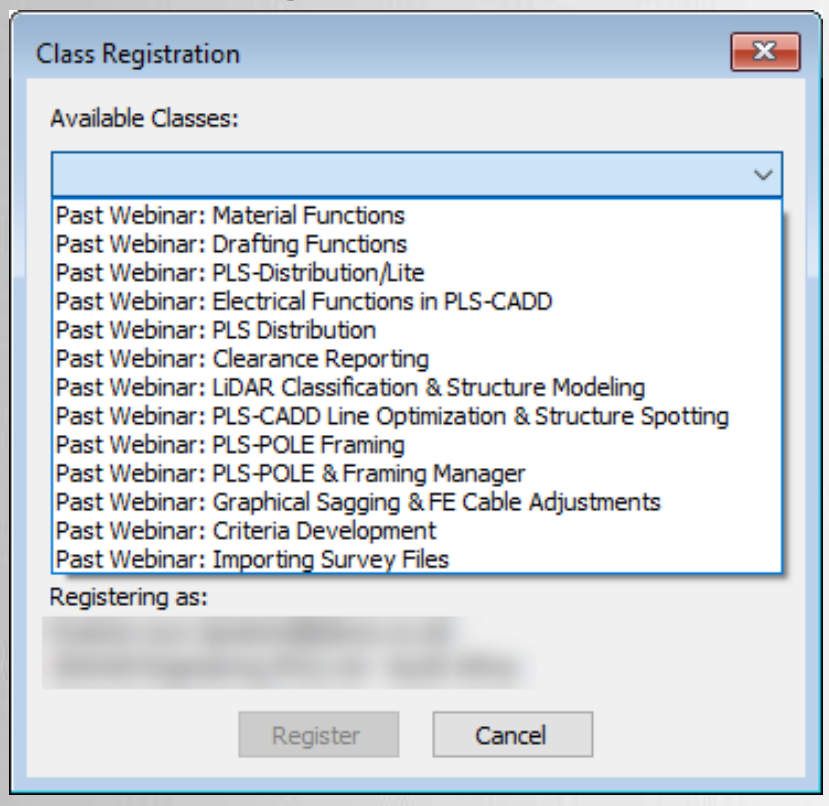

Outputs
10
Results reports
Load trees
A summary table of the loading on the Pole structure can be obtained by selecting Loads/ Loading Tree Report
A more graphical Load tree can be obtained as using the Drafting tools.
Section 8.5
Pole drawings
Scaled drawing sheets can be produced automatically by PLS-POLE according to user specified parameters. You have complete flexibility in choosing scales and page appearance.
The drawings can be sent directly to a Windows® printer, or they can be exported as DXF files readable by most CAD systems (AutoCAD ®, Microstation ®, etc.) for further enhancements, or they (together with other project documents) can automatically be converted to PDF documents as a truly universal way to share them with all interested parties.
Drawings can be embellished further by adding Inset Views and even attaching external files to be used.
Section 8

Webinars:

Power Line Systems produced a series of Webinars on selected topics during 2020.

These webinars are available through the latest version of the software by selecting:

Help/ Register for Training Classes…



In this dialog box you can access the Past Webinars. These are available in the drop down list.

You will receive a URL link to either watch the videos online, or to download them.

In total there is about 18 hours covering the 13 topics

Power Line Systems

IT'S ALL ABOUT YOUR POWER LINES

Advanced Sag & Tension
IEC
FAC 008/009
Materials Management
NESC
PLS-CADD ®
LiDAR Modeling
Structural Analysis
CSA
Pole Analysis
FAC 003
Distribution
CENELEC
Transmission
Line Ratings
Joint Use
IEEE
NERC Ratings
Line Optimization
Storm Hardening
Drafting
Project Estimating
Vegetation Management
1000+ Users in 100+ Countries
PLS-POLE
GO95
TOWER
ASCE

See www.powerlinesystems.com for more information
Contact info@powerlinesystems.com for additional information
Contact sales@powerlinesystems.com for quotation
Contact support@powerlinesystems.com for technical inquiries

POWER LINE® SYSTEMS

Madison, Wisconsin 53719, USA
Phone: 608- 238-2171
Fax: 608-238-9241
info@powline.com
www.powerlinesystems.com

POWER LINE® SYSTEMS

IT'S THE SOLUTION