Proposed workflow for TOWER

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

Proposed workflow for TOWER

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 TOWER 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:

Decision

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 TOWER

- 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
TOWER
Start a new project
Section 4.1
General project setup
Set Preferences
1
Load Master Files
2
General Menu
Define or Check the Component Libraries
3
Create Primary and Secondary Joints needed to model the structure
4
Define Tower Sections
5
Define Member Groups
6
Connect Angle or Round Members
7
Build the structures geometry by adding individual component elements
8
Attach Insulators and Clamps
9
Miscellaneous options
10
Link to PLS-CADD
11
Loads
12
Model Menu
13
Outputs

Optimization Options

TOWER has several optimization capabilities which can be used for both new or existing tower designs. The options that control optimization runs can be user specified.

This allows you to select an Alternate Grade of Steel, what bolts to consider for new members, what loads are used for bolt/connection design, how the program handles slenderness limit violations.

These options work in hand with the Optimize Cost factor entered in the **Components/Angle...** *Library as well as the selections made in the Member Groups table to control the scope and scale of optimization.*

General/Optimization Options...
Section 6

Post Processor Options

TOWER users are provided with the ability to post-process the program results.

The XML options are the best way to extract data from a TOWER run and the only ones supported for new applications.

General/Post Processor Options
Section 4.2.7

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*
  - *Leg compression graph and TIA/EIA 222 twist & out-of plumb graphs*

You can also create customised Member Labels, add a Warning to print in any reports, and suppress certain warnings in the 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/Non-linear) and Nonlinear convergence parameters.*
- *Structure shape (rectangular or triangular) used for drag coefficient selection.*
- *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 and results.

In TOWER there are also 3 additional tabs, the **Design Checks** *and* **Design Check Options** *tabs allow you to select which code/standard is used for the checks of the Member Strength, Connection Rupture, Crossing Diagonal, Included Angle, Climbing Load, Stub Angle.*

The **Redundant Members** *tab controls how redundant members are designed and checked in the model*

General/General Data
Section 4.2.1

General Menu:
The General Menu has many important settings

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.

For lattice towers this includes a Steel Material Library (.smp file), an Angle Library (.ang file), a Bolt Library (.bit file), a Cable Library (.cab file), an equipment Library (.eqp file) a Connection and Anchor Library (.can file) and an Insulator Library (.ins file).

If you make use of material tracking capabilities of PLS software then you can also load in 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

Define Tower Sections

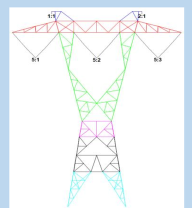
Tower models can be divided into sections. This allows you to globally assign loading or other parameters to all the members in one section.

For each section, you can assign separate drag or area adjustment factors to take care of various items which are not included in the model. Wind models are discussed in Section 5.1.3. *You can also import the calculated tower weight per section.*

Each section has a transverse and longitudinal face. Membership of sections to an individual face can either be automatic or manually controlled as discussed in Section 4.4.1.1.

You can also select a section as a basic module which can easily be replicated to build the model of a larger structure made of modular sub-assemblies.

Note: This step can be performed after all the members have been entered in the model, if desired, but must be performed before the analysis is run.



Geometry/Sections/Table Edit
Section 4.4

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 are sample 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

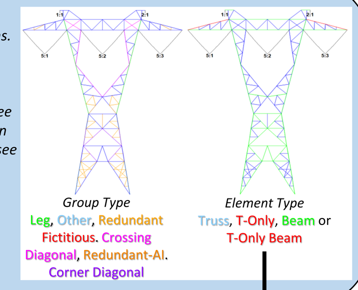
Define Member Groups

Angle groups are collections of angle or round members that have identical cross sections. A group is identified by its label, often called the "mark number", and its description. Angle Type, Size and Material Type is specified for each Group.

You further specify an **Element Type** *(truss members carry only axial loads, beams also see shear and moments see* Section 1.2.3*, and tension only members only function in tension see* Section 1.2.2*) and a* **Group Type** *(defines slenderness limits and other design checks see* Section 3.1.3.1*).*

Additionally you can chose how optimization will be performed for each Group. Analysis results are summarized by groups

Geometry/Groups/Table Edit
Section 4.5



Notes about Beam elements:

Bending moments do exist in real towers due to the detailing - but historically these have been accounted for with eccentricity/restraint concepts – i.e. lowering the capacity of members in compression and thereby eliminating the need to design for combined bending and compression.

The main purpose of using beam elements in a TOWER model is to **stabilize planar joints and mechanisms** *that occur when pin-jointed members connect. Beam members add stiffness to joints and can thus reduce the out of plane movement.*

Beam members are still axially loaded and reported moments are approximate only.

Excessive moments will generate a warning in TOWER reports and should be evaluated based on engineering judgement

Sections 1.2.3, 1.2.3.4 and 3.1.1.1

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 angle members, cables or guys. A CAN element can be attached to any joint in the model, and you may select up to four elements or two insulators to pass forces and moments to the CAN element. A CAN element has no graphical display and no physical attributes.

Geometry/Connections and Anchors...
Section 4.17

Add Equipment (from a Library):

Equipment can be added to a model to represent the various pieces of equipment placed on towers. 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 member.

Geometry/Equipment...
Section 4.9

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/Dead Loads and Drag Areas...
Section 4.10

Add Cables:

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

Geometry/Cables...
Section 4.8

Add Guy:

Guy wires are attached to structures to resist loads and prevent large deflections. Guys always connect to a existing joint and the other end is a ground anchor which is a program-generated fixed joint.

Geometry/Guys...
Section 4.7

Build the structures geometry by adding individual component elements

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

Foundation Footprint

For any joint connected to the ground in TOWER, you can enter either a Diameter for a circular foundation or a Length and Width for a rectangular foundation.

This lets you view the foundation footprint extents in both PLS-CADD and TOWER.

These footprints are used for Right of Way checks and are treated as part of the structure during Optimum Spotting when placing structures near spotting constraints.

Geometry/PLS-CADD/Foundation Footprint...
Section 4.14.1

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 8.2

Mapping Attachment Points to PLS-CADD Cable Sets

If you wish to use your structures 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 8.1

Family Manager

The Family Manager in TOWER allows for all body and leg extensions configurations to be saved in one *.TOW *file. For this to work, all towers must have the same top geometry (the common body) and only differ based on the body/leg extensions. The Family Manager will divide the model into reusable pieces and allow you to quickly create multiple configurations by stacking body and/or leg extensions beneath the common body.*

You can analyze these configurations together and TOWER will automatically generate a Group Summary Super Set and a Foundation Reaction Super Set.

For Family Manager to work, all component files, angle groups, sections, insulators, guys and equipment are shared among all configurations

This tool can be used powerfully when combined with PLS-CADD and the **Optimum Body and Leg Extension Selection** *tool that finds the best tower configuration to fit the ground surface.*

Model/Family Manager
Section 6.4

Optimization

TOWER has a few built-in automatic design or optimization routines. These tools offer an Engineer the opportunity to automatically select the member sizes and number of bolts, in an attempt to reinforce a few members of an existing tower OR to optimize the full tower in one automated process to find the most economical solution for new towers.

This can be for faster and more accurate than a simple trial and error approach which is the alternative.

The optimization routines that are available are:

- **Model/Interactive Member Sizing...**
- **Model/Optimize...**
- **Model/Auto-Fix Angles...**

Sections 6.1, 6.2 and 6.3

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 (labelled* **Analysis Results***) and*
- *a short report (labelled* **Summary Results***)*

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

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.*

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 TOWER, or can be saved as Text or PDF files.

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/JSON file for outside post-processing.

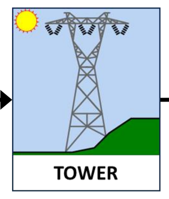
TOWER automates the following design checks:

- *Member Compression (L/r)*
- *Member Tension (net section)*
- *Connection shear (comp. & tension forces)*
- *Connection bearing (comp.& tension forces)*
- *Redundant members*
- *Connection rupture (edge, end & spacing distances & block shear)*
- *Crossing & Corner diagonals (out-of-plane buckling check)*
- *Included Angle*
- *Climbing Load*
- *Stub Angle*

Section 2.4
Calculations from Section 3.1.3

Create a project directory

Open



Start a new project
Section 4.1

General project setup

1 Set Preferences

Load Master Files

2 General Menu

Define or Check the Component Libraries

3 Create Primary and Secondary Joints needed to model the structure

4 Define Tower Sections

5 Define Member Groups

6 Connect Angle or Round Members

7 Build the structures geometry by adding additional structural elements

8 Attach Insulators and Clamps

9 Miscellaneous options

10 Link to PLS-CADD

11 Loads

12 Model Menu

13 Outputs

Create Primary and Secondary Joints needed to model the structure

All members added to the structure model need to be placed between 2 joints. The joints are node points between members and points of load transfer.

*It is good practice to develop a "tower coding sheet" which is a drawing showing the joint names and symmetry codes and have a consistent joint naming convention. (*Fig. 4.1-1 *in the manual)*
Utilize symmetry about the X and Y axes to minimize the number of joints to be entered.

Each joint has:

- *3 degrees of freedom for joint displacement (XYZ)*
- *3 degrees of freedom for joint rotation (XYZ)*

Most joints are usually ***Free****, and usually only the base joints are* ***Fixed*** *in displacement and rotation.*

Section 4.3

Primary Joints

Primary joints are defined by specifying:

- *The X, Y, Z coordinates*

Primary joints are usually used where there is a change of slope or at the tips or arms of the top of structures. Try to use as few as possible

Geometry/Primary Joints...
Section 4.3.1

Secondary Joints

These joints are defined relative to other joints (these can be primary or other secondary joints).

Secondary joints are defined by specifying:

- *The Origin and End joint that it is relative to,*
- *Either a fractional distance or elevation between the other joints*

Geometry/Secondary Joints...
Section 4.3.2

Connect Angle or Round Members

Use the tools in the ***Geometry/Members/*** *menu*

Add Members:

*This can be done in a tabular format by directly entering data in the Angle Member connectivity table (***Geometry/Members/Table Edit...***) but is easiest to do in a graphical mode and clicking between the relevant joints (***Geometry/Members/Add...***).*

This can be the most time-consuming part of modelling a tower and requires deep insight into structural modelling for each angle element on the tower and knowledge of applied design code provisions. But is also critically important to the overall accuracy of the model.

Input needed is:
Group, Section, Symmetry Code, Origin and End Joints, Eccentricity and Restraint Code or Curve numbers for compression design, Unbraced Length Ratios RLX, RLY, and RLZ, Bolt type, # of bolts, # of bolt holes, # of shear planes, indication which leg is connected, connection information for rupture design (Short and Long edge distance, end distance, bolt spacing, shear path length and tension path length, restraint coefficient and an optional connection modifier).

Using symmetry can dramatically speed up modelling.
Section 4.6.1

Capacities and overrides:

TOWER calculates the member design strength based on the selected design code/standard. However, there may be instances where you want to specify that member strength yourself. This maybe because the program is not able to handle a special situation (i.e. your member is not an angle or round member), or you wish to override the built-in strength calculations.

Geometry/Members/Capacities and Overrides...
Section 4.6.4

Splitting Existing Members into Additional Joints and Members

TOWER allows you to divide or split existing/defined members into portions.
This can be useful since it saves you from manually adding joints in and allows for the joint locations to be accurately placed in the model.

It can be useful to change the default TOWER created joint and member name and follow your naming convention, this would be done with the **Geometry/Members/Prompt For Member Label On Creation** *and* **Geometry/Prompt For Joint Label On Creation** *options.*

Geometry/Members/Split...
Section 4.6.2

Connecting a Pair of Crossing Members at their Common Point

It is generally not a good idea to make a joint at the intersection of two diagonals on a TOWER face or in a horizontal diaphragm. However, there are some cases where you will want to have a joint at these intersections, for example to attach some redundant or other members. The graphical function will automatically create a joint at the intersection of the selected members and split these members into two so that they meet at the new joint.

It can be useful to change the default TOWER created joint and member name and follow your naming convention, this would be done with the **Geometry/Members/Prompt For Member Label On Creation** *and* **Geometry/Prompt For Joint Label On Creation** *options.*

Geometry/ Members/ Intersect
Section 4.6.3

Find Allowable Swing Angles

TOWER is capable of calculating the allowable swing 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 4.11.1

Calculating Allowable Load Angles for 2-part Insulators

TOWER 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 4.11.1

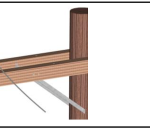
Attach Insulators and Clamps:

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 tools in the **Geometry/Insulators/**
Section 4.11

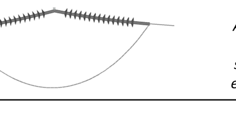
Add Clamp:

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 weight but has strength properties.*



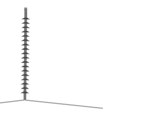
Add Strain:

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



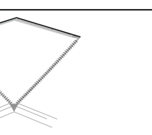
Add Suspension:

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



Add 2-Part:

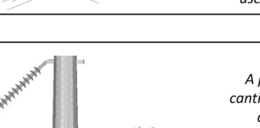
A 2-part insulator normally supports a conductor at the junction point of two insulators. The most common 2-part insulators are V-strings, but they can be used to model hangers or pivoting posts.



Add Post:

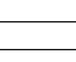
A post insulator is modelled as a short, cantilevered beam element rigidly attached at its origin to the structure model.

- *Orientation can be horizontal, vertical or upswept (inclined).*
- *May optionally include brace elements.*



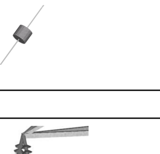
Add Guy Strain:

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



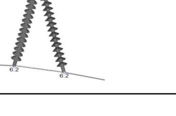
Add Lambda:

A Lambda insulator supports a conductor at two locations and is attached to a structure at one, sometimes referred to as an Inverted V-string.
These are used to improve reliability at road crossings and unequal span loading.



Add Double Pin:

A Double Pin insulator supports a wire at two locations and is attached to a structure at two locations. Double Pin insulators are rigid, typically point up and are often used on distribution structures.



Add Double Suspension:

A Double Suspension insulator supports a wire at two locations and is attached to a structure at two locations. They are flexible, point down and are sometimes used to increase structure clearances to lattice towers at line angles or to add redundancy.

Foundation Strength:

You can enter both the Strength (capacities) and Stiffness data for all joints with at least one restraint in X-, Y-, or Z-directions for TOWER models.
The actual foundation is not modelled, but the capacity of the foundation system can be checked by TOWER during the analysis.

Geometry/Foundation Strength....
Section 4.12

Dead Loads and Drag Areas:

If you wish to account for vertical and wind loads on permanently attached equipment or fixtures which you aren't modelling, you can do so by specifying A Vertical Dead Load or Wind Area to any joint in the model.

Geometry/Dead Loads and Drag Areas...
Section 4.10

Linear Appurtenances:

When modelling Communication towers (EIA load cases), it is possible to model linear appurtenances to simulate and automate the effects of dead, wind and ice loading from conduits, ladders, conductors, etc attached to the structure over a certain elevation range.

Geometry/Linear Appurtenances...
Section 4.13

Loads

Design loads on transmission towers 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 TOWER is run for transmission/distribution structures in design check mode, or in one of the allowable span modes (***Sections 1.1.4 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 tabular 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.

Loads/Batch Import or Edit Concentrated Loads...
Section 5.3.1

Results reports

Load trees

A summary table of the loading on the TOWER structure can be obtained by selecting **Loads/Loading Tree Report**
Section 8.5.2

A more graphical Load tree can be obtained as using the Drafting tools.
Drafting/ Add Loading Tree Report... *and* **Drafting/ Add Loading Tree Views...**
Sections 8.5.1 and 8.5.2

Tower drawings

Scaled drawing sheets can be produced automatically by TOWER 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 for tabulated results, section cuts, load trees or reports, material lists and member group reports and even attaching external files to be used.

Section 8

Imperial (U.S.)

Metric (S.I.)

What Unit System should be used?

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

File and Directory Mapping

The location of default locations for the directories or library files is mapped here.

*For lattice towers this includes a Steel Material Library (**.smp** file), an Angle Library (**.ang** file), a Bolt Library (**.blt** file), a Cable Library (**.cab** file), an equipment Library (**.eqp** file) a Connection and Anchor Library (**.can** file) and an Insulator Library (**.inl** file).*

If you make use of material tracking capabilities of PLS software then you can also load in 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.**

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/Non-linear) and Nonlinear convergence parameters.*
- *Structure shape (rectangular or triangular) used for drag coefficient selection.*
- *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 and results.

*In TOWER there are 3 additional tabs, the **Design Checks** and **Design Check Options** tabs allow you to select which code/standard is used for the checks of the* Member Strength, Connection Rupture, Crossing Diagonal, Included Angle, Climbing Load, Stub Angle.
*The **Redundant Members** tab controls how redundant members are designed and checked in 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*
  - *Leg compression graph and TIA/EIA 222 twist & out-of plumb graphs*

You can also create customised Member Labels, add a Warning to print in any reports, and suppress certain warnings in the reports.

General/Output Options
Section 4.2.2

Post Processor Options

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

General/Post Processor Options
Section 4.2.7

Optimization Options

TOWER has several optimization capabilities which can be used for both new or existing tower designs. The options that control optimization runs can be user specified.

*This allows you to select an Alternate Grade of Steel, what bolts to consider for new members, what loads are used for bolt/connection design, how the program handles slenderness limit violations. These options work in hand with the Optimize Cost Factor entered in the **Components/Angle...** Library as well as the selections made in the Member Groups table to control the scope and scale of optimization.*

General/Optimization Options...
Section 6

Create Primary and Secondary Joints needed to model the structure

All members added to the structure model need to be placed between 2 joints. The joints are node points between members and points of load transfer.

It is good practice to develop a "tower coding sheet" which is a drawing showing the joint names and symmetry codes and have a consistent joint naming convention. (Fig. 4.1-1 in the manual)
Utilize symmetry about the X and Y axes to minimize the number of joints to be entered.

Each joint has:

- *3 degrees of freedom for joint* displacement *(XYZ)*
- *3 degrees of freedom for joint* rotation *(XYZ)*

Most joints are usually **Free***, and usually only the base joints are* **Fixed** *in displacement and rotation.*

Section 4.3

3

Primary Joints

Primary joints are defined by specifying:

- *The X, Y, Z coordinates*

Primary joints are usually used where there is a change of slope or at the tips or arms of the top of structures. Try to use as few as possible

Geometry/Primary Joints...

Section 4.3.1

Secondary Joints

These joints are defined relative to other joints (these can be primary or other secondary joints).

Secondary joints are defined by specifying:

- *The Origin and End joint that it is relative to,*
- *Either a fractional distance or elevation between the other joints*

Geometry/Secondary Joints...

Section 4.3.2

4

Define Tower Sections

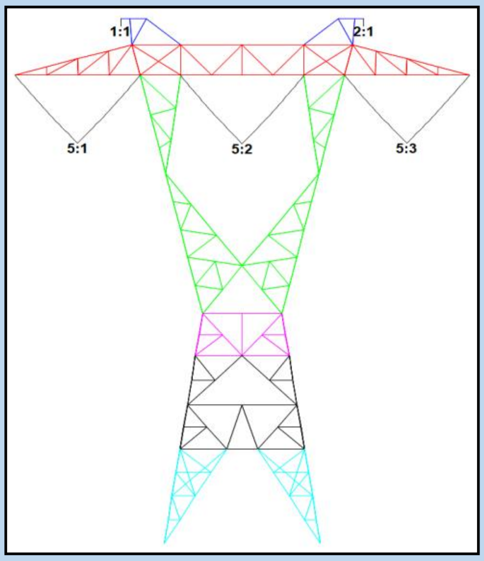
Tower models can be divided into sections. This allows you to globally assign loading or other parameters to all the members in one section.

For each section, you can assign separate drag or area adjustment factors to take care of various wind models or the effects of redundant members not included in the model. Wind models are discussed in Section 5.1.3. You can also adjust the calculated tower weight per section.

Each section has a transverse and longitudinal face. Membership of section members into its face can either be automatic or manually controlled as discussed in Section 4.4.1.1.

You can also select a section as a basic module which can easily be replicated to build the model of a larger structure made of modular sub-assemblies.

Note: This step be can be performed after all the members have been entered in the model, if desired, but must be performed before the analysis is run.



Geometry/Sections/Table Edit
Section 4.4

5

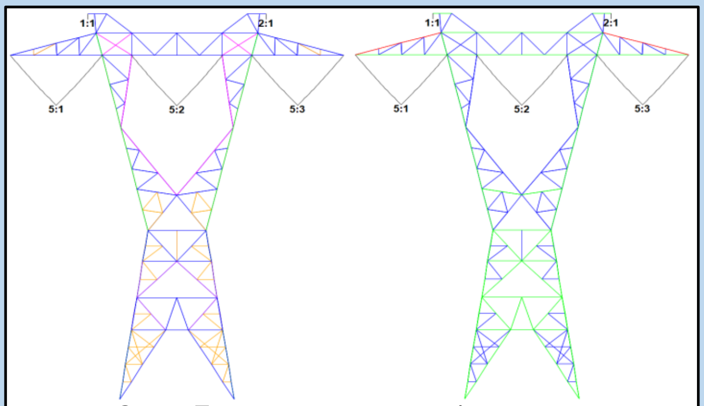
Define Member Groups

Angle groups are collections of angle or round members that have identical cross sections.
A group is identified by its label, often called the "mark number", and its description.
Angle Type, Size and Material Type is specified for each Group.

*You further specify an **Element Type** (truss members carry only axial loads, beams also see shear and moments see Section 1.2.3, and tension only members only function in tension see Section 1.2.2) and a **Group Type** (defines slenderness limits and other design checks see Section 3.1.3.1).*

Additionally you can chose how optimization will be performed for each Group.
Analysis results are summarized by groups

Geometry/Groups/Table Edit
Section 4.5



Group Type
Leg, Other, Redundant Fictitious, Crossing Diagonal, Redundant-AI, Corner Diagonal

Element Type
Truss, T-Only, Beam or T-Only Beam

Notes about Beam elements:
Bending moments do exist in real towers due to the detailing - but historically these have been accounted for with eccentricity/restraint concepts – i.e. lowering the capacity of members in compression and thereby eliminating the need to design for combined bending and compression.

*The main purpose of using beam elements in a TOWER model is to **stabilize planar joints and mechanisms** that occur when pin-jointed members connect**.** Beam members add stiffness to joints and can thus reduce the out of plane movement.*

Beam members are still axially loaded and reported moments are approximate only.

Excessive moments will generate a warning in TOWER reports and should be evaluated based on engineering judgement
Sections 1.2.3, 1.2.3.4 and 3.1.1.1

6

Connect Angle or Round Members

Use the tools in the ***Geometry/Members/*** *menu*

Add Members:

*This can be done in a tabular format by directly entering data in the Angle Member connectivity table (***Geometry/Members/Table Edit...***) but is easiest to do in a graphical mode and clicking between the relevant joints (***Geometry/Members/Add...***).*

This can be the most time-consuming part of modelling a tower and requires deep insight into structural modelling for each angle element on the tower and knowledge of applied design code provisions. But is also critically important to the overall accuracy of the model.

Input needed is:
Group, Section, Symmetry Code, Origin and End Joints, Eccentricity and Restraint Code or Curve numbers for compression design, Unbraced Length Ratios RLX, RLY, and RLZ, Bolt type, # of bolts, # of bolt holes, # of shear planes, indication which leg is connected, connection information for rupture design (Short and Long edge distance, end distance, bolt spacing, shear path length and tension path length, restraint coefficient and an optional connection modifier).

Using symmetry can dramatically speed up modelling.
Section 4.6.1

Capacities and overrides:

TOWER calculates the member design strength based on the selected design code/standard. However, there may be instances where you want to specify that member strength yourself. This maybe because the program is not able to handle a special situation (i.e. your member is not an angle or round member), or you wish to override the built-in strength calculations.

Geometry/Members/Capacities and Overrides...
Section 4.6.4

Splitting Existing Members into Additional Joints and Members

TOWER allows you to divide or split existing/defined members into portions.
This can be useful since it saves you from manually adding joints in and allows for the joint locations to be accurately placed in the model.

It can be useful to change the default TOWER created joint and member name and follow your naming convention, this would be done with the **Geometry/Members/Prompt For Member Label On Creation** *and* **Geometry/Prompt For Joint Label On Creation** *options.*

Geometry/Members/Split...
Section 4.6.2

Connecting a Pair of Crossing Members at their Common Point

It is generally not a good idea to make a joint at the intersection of two diagonals on a TOWER face or in a horizontal diaphragm. However, there are some cases where you will want to have a joint at these intersections, for example to attach some redundant or other members. The graphical function will automatically create a joint at the intersection of the selected members and split these members into two so that they meet at the new joint.

It can be useful to change the default TOWER created joint and member name and follow your naming convention, this would be done with the **Geometry/Members/Prompt For Member Label On Creation** *and* **Geometry/Prompt For Joint Label On Creation** *options.*

Geometry/ Members/ Intersect
Section 4.6.3

7

Build the structures geometry by adding individual component elements

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

Add Guy:
Guy wires are attached to structures to resist loads and prevent large deflections. Guys always connect to a existing joint and the other end is a ground anchor which is a program-generated fixed joint.

Geometry/Guys...
Section 4.7

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

Geometry/Cables...
Section 4.8

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/Dead Loads and Drag Areas...
Section 4.10

Add Equipment (from a Library):
Equipment can be added to a model to represent the various pieces of equipment placed on towers. 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 member.
Geometry/Equipment...
Section 4.9

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 angle members, cables or guys. A CAN element can be attached to any joint in the model, and you may select up to four elements or two insulators to pass forces and moments to the CAN element.
A CAN element has no graphical display and no physical attributes.

Geometry/Connections and Anchors...
Section 4.17

POWER LINE SYSTEMS ®

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.

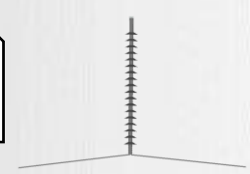
Use the tools in the **Geometry/Insulators/**

Section 4.11

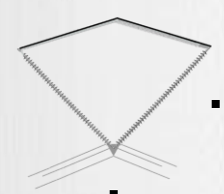
Add Clamp:
A clamp is a device that transfers concentrated loads from ground wires, conductors or other sources to an attachment point on the structure. A clamp has ***no*** *physical dimension and weight but has strength properties.*

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



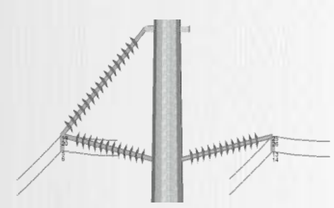
Add 2-Part:
A 2-part insulator normally supports a conductor at the junction point of two insulators. The most common 2-part insulators are V-strings, but they can be used to model hangers or pivoting posts.



Find Allowable Swing Angles
TOWER is capable of calculating the allowable swing 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 4.11.1

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

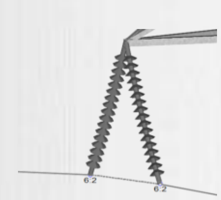


Add Guy Strain:
Guy strain insulators are applied to down guys to provide insulation from energized portions of the other insulators or cables in PLS-CADD.

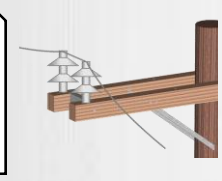
Calculating Allowable Load Angles for 2-part Insulators
TOWER can determine the allowable load angles for 2-part insulators.
The allowable load angles will guarantee that neither side of the 2-part insulator will ever go into compression.

Geometry/Insulators/Calculate Allowable 2-Part Load Angles
Section 4.11.1

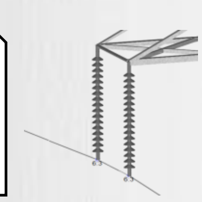
Add Lambda:
A Lambda insulator supports a conductor at two locations and is attached to a structure at one, sometimes referred to as an Inverted V-string.
These are used to improve reliability at road crossings and unequal span loading.



Add Double Pin:
A Double Pin insulator supports a wire at two locations and is attached to a structure at two locations. Double Pin insulators are rigid, typically point up and are often used on distribution structures.



Add Double Suspension:
A Double Suspension insulator supports a wire at two locations and is attached to a structure at two locations. They are flexible, point down and are sometimes used to increase structure clearances to lattice towers at line angles or to add redundancy.



Miscellaneous options

9

Foundation Properties:
You can enter both the Strength (capacities) and Stiffness data for all joints with at least one restraint in X-, Y-, or Z-directions for TOWER models.
The actual foundation is not modelled, but the capacity of the foundation system can be checked by TOWER during the analysis.

Geometry/Foundation Strength....
Section 4.12

Dead Loads and Drag Areas:
If you wish to account for vertical and wind loads on permanently attached equipment or fixtures which you aren't modelling, you can do so by specifying A Vertical Dead Load or Wind Area to any joint in the model.

Geometry/Dead Loads and Drag Areas...
Section 4.10

Linear Appurtenances:
When modelling Communication towers (EIA load cases), it is possible to model linear appurtenances to simulate and automate the effects of dead, wind and ice loading from conduits, ladders, conductors, etc. attached to the structure over a certain elevation range.

Geometry/Linear Appurtenances...
Section 4.13

Link to PLS-CADD

10

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

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

Foundation Footprint

For any joint connected to the ground in TOWER, you can enter either a Diameter for a circular foundation or a Length and Width for a rectangular foundation.

This lets you view the foundation footprint extents in both PLS-CADD and TOWER.

These footprints are used for Right of Way checks and are treated as part of the structure during Optimum Spotting when placing structures near spotting constraints.

Geometry/PLS-CADD/Foundation Footprint...

Section 4.14.1

Loads

11

Loads

Design loads on transmission towers 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 TOWER is run for transmission/ distribution structures in design check mode, or in one of the allowable span modes (**Sections 1.1.4 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.

Loads/Batch Import or Edit Concentrated Loads...
Section 5.3.

Model Menu

12

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 (labelled **Analysis Results**) and*
- *a short report (labelled **Summary Results**)*

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

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 TOWER, or can be saved as Text or PDF files.

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/JSON file for outside post-processing.

TOWER automates the following design checks:

- *Member Compression (L/r)*
- *Member Tension (net section)*
- *Connection shear (comp. & tension forces)*
- *Connection bearing (comp.& tension forces)*
- *Redundant members*
- *Connection rupture (edge, end & spacing distances & block shear)*
- *Crossing & Corner diagonals (out-of-plane buckling check)*
- *Included Angle*
- *Climbing Load*
- *Stub Angle*

Section 2.4

***Calculations from* Section 3.1.3**

Optimization

TOWER has a few built-in automatic design or optimization routines. These tools offer an Engineer the opportunity to automatically select the member sizes and number of bolts, in an attempt to reinforce a few members of an existing tower OR to optimize the full tower in one automated process to find the most economical solution for new towers.

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/Interactive Member Sizing…**
- **Model/Optimize…**
- **Model/Auto-Fix Angles…**

Sections 6.1, 6.2 and 6.3

Family Manager

*The Family Manager in TOWER allows for all body and leg extensions configurations to be saved in one *.TOW file. For this to work, all towers must have the same top geometry (the common body) and only differ based on the body/leg extensions. The Family Manager will divide the model into reusable pieces and allow you to quickly create multiple configurations by stacking body and/or leg extensions beneath the common body.*

You can analyze these configurations together and TOWER will automatically generate a Group Summary Super Set and a Foundation Reaction Super Set.

For Family Manager to work, all component files, angle groups, sections, insulators, guys and equipment are shared amongst all configurations

This tool can be used powerfully when combined with PLS-CADD and the **Optimum Body and Leg Extension Selection** *tool that finds the best tower configuration to fit the ground surface.*

Model/Family Manager

Section 6.4

POWER LINE SYSTEMS

Outputs

13

Results reports

Load trees

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

Section 8.5.2

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

Drafting/ Add Loading Tree Report... *and* **Drafting/ Add Loading Tree Views...**

Sections 8.5.1 and 8.5.2

Tower drawings

Scaled drawing sheets can be produced automatically by TOWER 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 for tabulated results, section cuts, load trees or reports, material lists and member group reports and even attaching external files to be used.

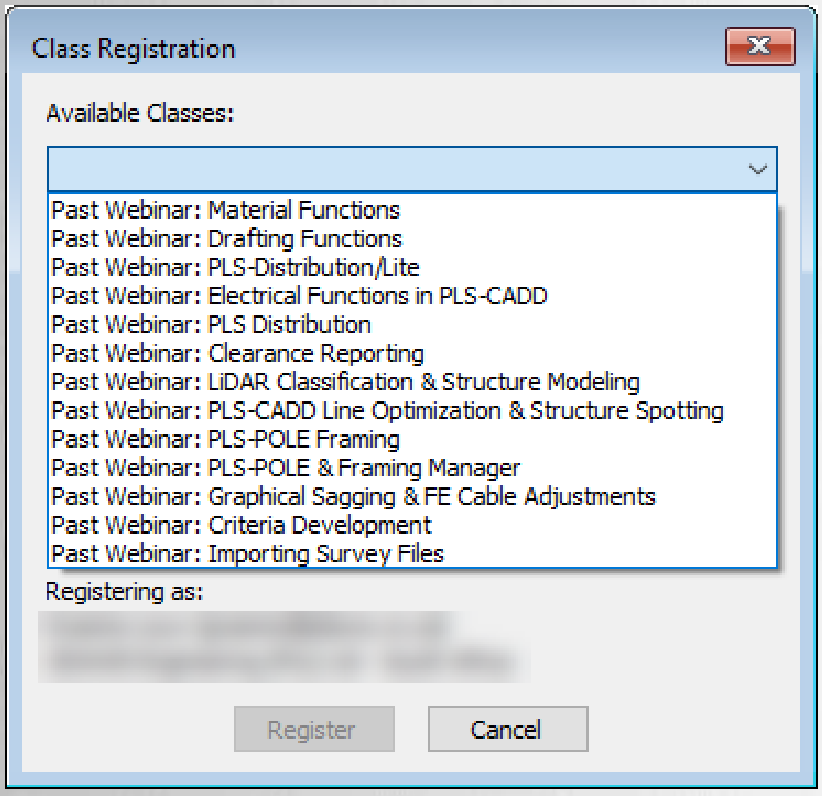
Section 8

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
NERC Ratings
Line Optimization
IEEE
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 sales@powerlinesystems.com for quotation
Contact support@powerlinesystems.com for technical inquiries

POWER LINE® SYSTEMS
Part of Bentley Systems
Madison, Wisconsin 53719, USA
Phone: 608- 238-2171
Fax: 608-238-9241
support@powerlinesystems.com
www.powerlinesystems.com