TowerSoft
ENGINEERING SOFTWARE

# TSTOWER FOR POLES

## STRUCTURAL ANALYSIS SOFTWARE FOR COMMUNICATION POLES

## USER'S MANUAL


BY

*JOHN WAHBA, PH.D., P.ENG., P.E.*

*MATTHEW MALINOWSKI, P.ENG.*



WWW.TOWERSFT.COM

# TABLE OF CONTENTS

# Chapter 1 INTRODUCTION

***TSTower*** is a general computer program for the analysis and design of cantilevered tubular pole structures. Poles can have a round or multiple sided cross sections. The program will perform the analysis using the United States applicable code TIA/EIA 222-F, TIA/EIA-222-G (Draft) or the Canadian equivalent CSA S37-94 and CSA S37-01. Load generation, capacity assessments follow the applicable codes, and the structural analysis is based on a three-dimensional beam model.

The program analyzes the pole for all loads and determines the capacity of the sections. The results are displayed graphically on the screen. This allows the designer to visually check the adequacy of the design and quickly make adjustments to achieve the optimum solution. The program allows for a full printout or a summary printout of the results. A graphical printout of the profile is also available.

## *HARDWARE REQUIRMENTS*

The following minimum system requirements to run TSTower:

- An IBM compatible PC equipped with a Pentium processor running under Windows 98, 2000, NT or XP
- 30 MB free disk space
- 16 MB RAM
- 14 inch SVGA monitor with (800 x 600 min resolution)
- A pointing device, a mouse or graphical tablet configured to work under windows.
- An optional printer that is set from windows

## *INSTALLING TSTOWER*

The TSTower installation is initiated from windows. From start menu choose run, and choose file setup.exe

The setup program will create a TSTower directory on the hard drive at a location designated by the user. The user may change the name of the folder in which the program will copy all necessary files required to run TSTower.

Choose the folder name and click OK, the installation program will copy all necessary files into the different directories of your system. Follow the program instructions and place the subsequent disks into the disk drive.

At the end of the installation the program will notify you that the installation was completed successfully. Refer to the purchasers agreement regarding the number of authorized users allowed to run the program.

## *DISCLAIMER*

Extensive care has been taken during the development and testing of TSTower program to ensure that both the source code and the underlying engineering principles comply with standard engineering practice. Should any discrepancies or possible program errors occur, please notify TowerSoft immediately.

> TOWERSOFT DISCLAIMS ALL WARRANTIES IMPLIED OR OTHERWISE WITH REGARDS TO THE SOFTWARE. BY USING THE SOFTWARE, THE USER AGREES THAT NEITHER TOWERSOFT NOR ITS EMPLOYEES SHALL BE LIABLE FOR ANY LOSS, DAMAGE, OR EXPENSE OF ANY KIND WHICH IS CAUSED DIRECTLY OR INDIRECTLY BY THE USE, PERFORMANCE, MAINTAINANCE, SERVICE OR CONDITION OF THE SOFTWARE. IN NO EVENT WILL TOWERSOFT BE LIABLE FOR SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGE RESULTING FROM USE OF THIS SOFTWARE.

User's comments and suggestions are welcomed. Please forward all your comments to support@towersft.com.

# Chapter 2 INPUT

***TSTower*** is an analysis software program. The user inputs all the necessary information required to perform the analysis, the finite element program is run, and the results are shown in a graphical format on the screen.

For design purpose, the user decides a tentative section and analysis is performed. The user examines the results of the preliminary selection and modifies any of the design parameters before reanalyzing the structure. Using successive runs, the user can quickly arrive at an optimum solution. The designer makes the choices, interprets the output and has full control on the design process.

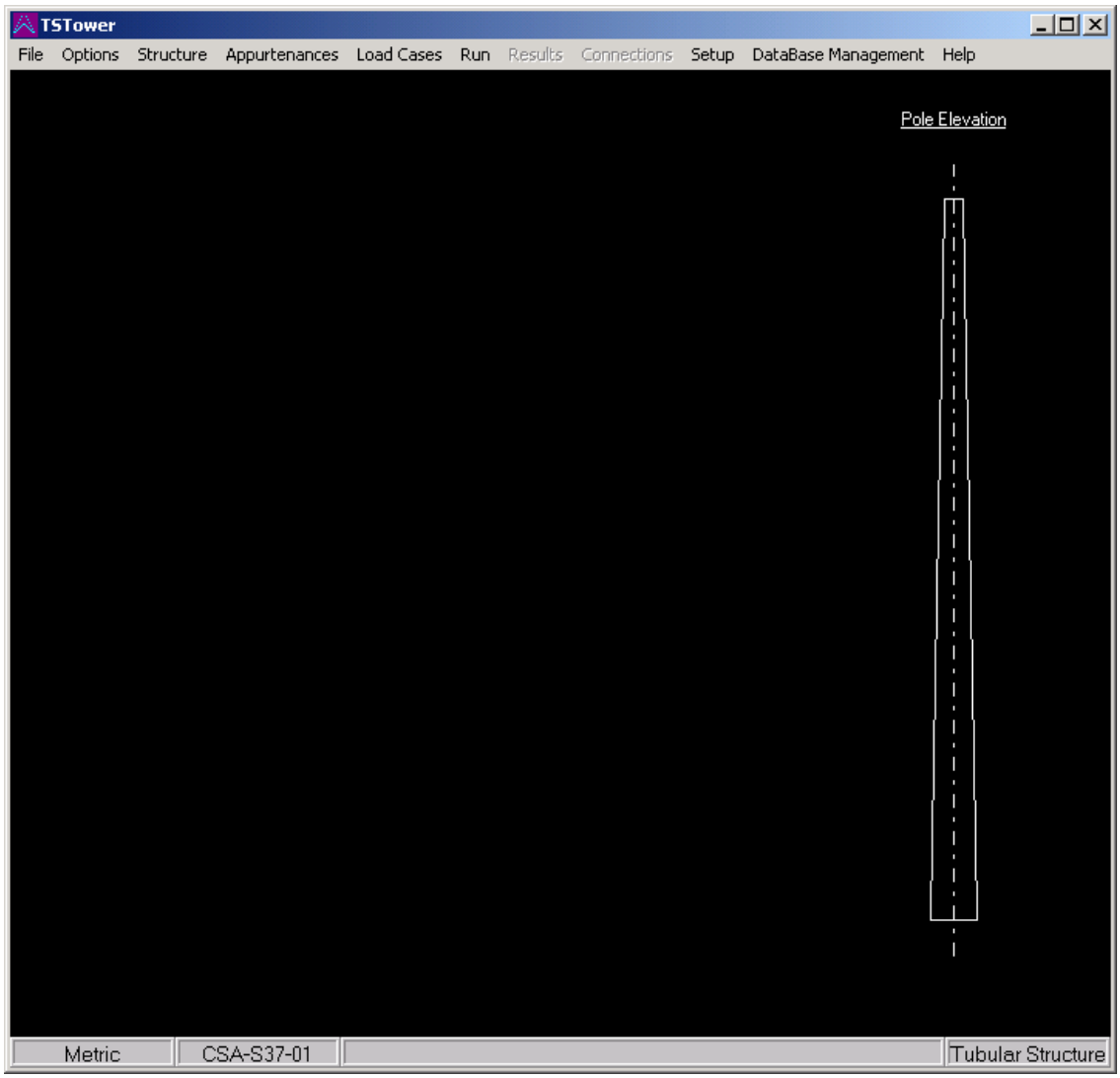
This Chapter illustrates how ***TSTower*** work by explaining the input of a typical example. The input values and results of the example are shown on the figures in the chapter. This example may not use all the features of the software, however it gives the user a quick introduction to the main features and the use of the program. Features available in the program that are not used for this example are explained in the context.

## *EXAMPLE*

This example is based on a 131.23ft pole, made of 6 sections, with (2) microwave dishes at an elevation of 98ft and PCS antennas at the top of the pole. The structure will be analyzed loading as per EIA-222-F.

### START THE PROGRAM

When TSTower starts, the data will be initialized with default settings. The screen will be blank as shown.

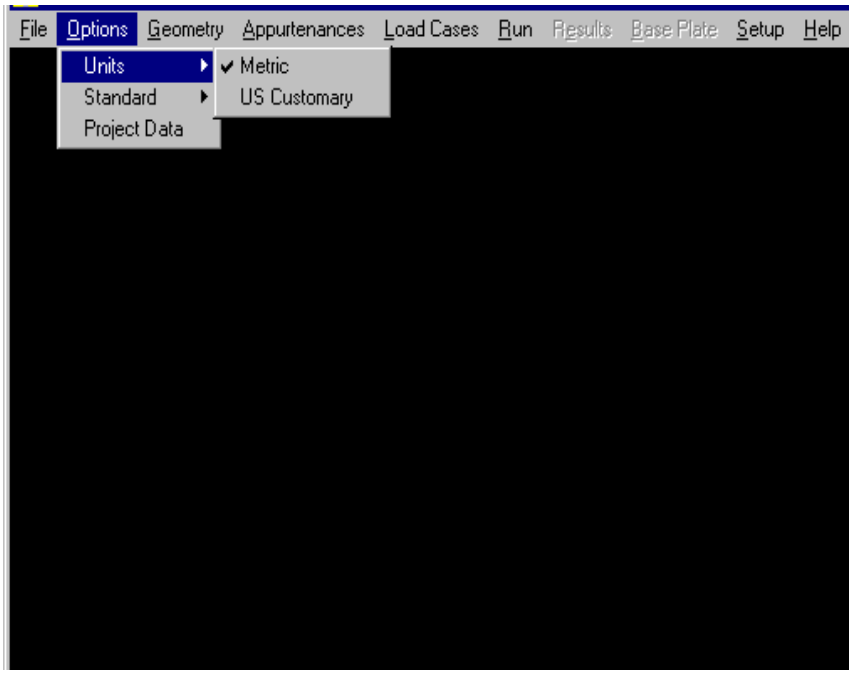



Note the status bar at the bottom of the windows showing the following data in order:

1- The current mode of units (metric or imperial)

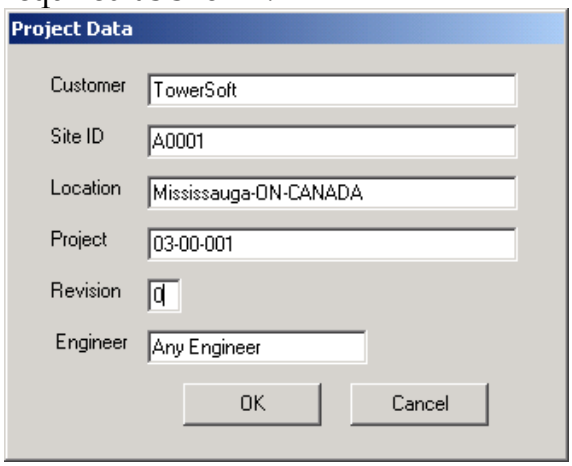
2- The applicable code of design (CSA S37-01, CSA S37-94, EIA 222-F, EIA 222-G (Draft))
3- The problem file name, and path

Select **Options** from the main menu and from the list choose the required code and the system of units.




## PROJECT DEFINITION

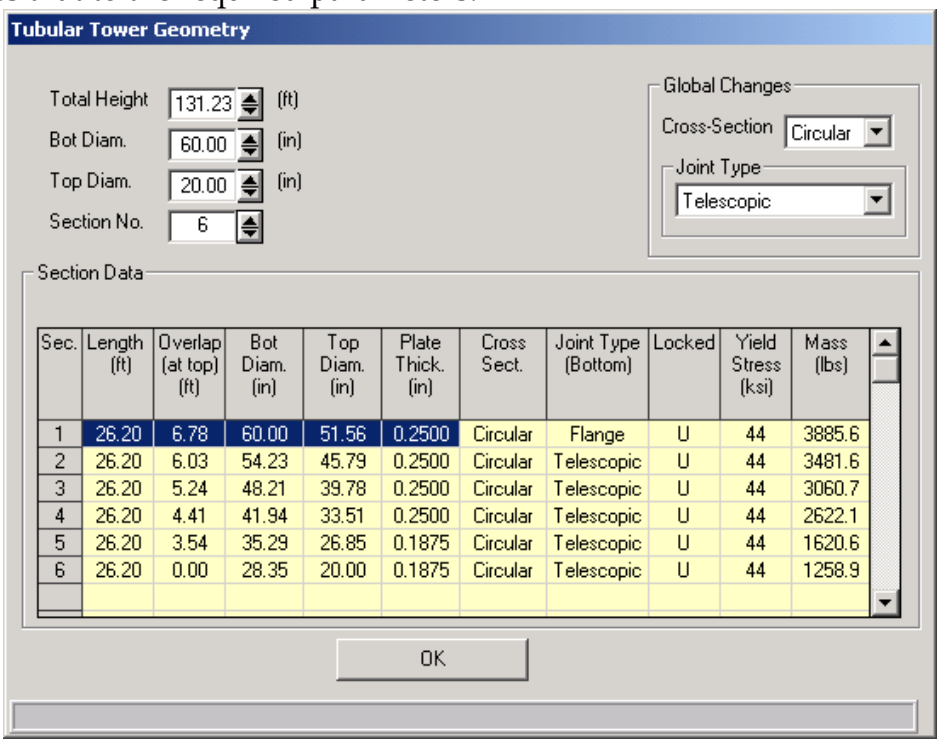
Select **Project Data** from the **Options** menu. Type in any identification data required as shown.



To add data to any box, move the cursor into the box, click inside the box and type in the data using the keyboard. To change any existing data in an edit box, delete the entry using the <Del> key or the <Backspace> key. Alternatively, highlight the existing characters by pressing and dragging the left mouse button, then type over the existing entry. This is a standard Windows feature.

## GEOMETRY DEFINITION

Select **Geometry** from the main menu, this will show the Geometry Definition Window. Initially the window will show default geometry data and the user changes that to the required parameters.




| Sec. | Length (ft) | Overlap (at top) (ft) | Bot Diam. (in) | Top Diam. (in) | Plate Thick. (in) | Cross Sect. | Joint Type (Bottom) | Locked | Yield Stress (ksi) | Mass (lbs) |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 26.20 | 6.78 | 60.00 | 51.56 | 0.2500 | Circular | Flange | U | 44 | 3885.6 |
| 2 | 26.20 | 6.03 | 54.23 | 45.79 | 0.2500 | Circular | Telescopic | U | 44 | 3481.6 |
| 3 | 26.20 | 5.24 | 48.21 | 39.78 | 0.2500 | Circular | Telescopic | U | 44 | 3060.7 |
| 4 | 26.20 | 4.41 | 41.94 | 33.51 | 0.2500 | Circular | Telescopic | U | 44 | 2622.1 |
| 5 | 26.20 | 3.54 | 35.29 | 26.85 | 0.1875 | Circular | Telescopic | U | 44 | 1620.6 |
| 6 | 26.20 | 0.00 | 28.35 | 20.00 | 0.1875 | Circular | Telescopic | U | 44 | 1258.9 |

In this window the user inputs the total height, top diameter of the pole, bottom diameter and number of sections along the height of the pole (for multi-sided poles, diameter is considered as the outside dimension between two opposite flat sides)

Note that the bottom diameter of the pole (or any section of it) cannot be specified to be less than the top diameter.

Also on the same window the user specifies the number of sides of the cross-section and the connection type (splice or telescopic).
The following values are the limits for these input fields:

- Total Height (5m - 150m)
  (10ft – 400ft)
- Top Diameter (102mm – 3000mm)
  (4in – 120in)
- Bottom Diameter (102mm – 3000mm)
  (4in – 120in)
- Number of Sections (1 – 10)
  *Also, a minimum of 1.0m (3ft) for section length is imposed*

- Diameter ratio (r) for poles with 16 sides or greater. This is used for calculation of the wind loads as per EIA-222-F Table 2

Cross-section type and Joint Type can be set for entire structure in "Global Changes" box. If this feature is used it will overwrite all previous entries in "Section Data". Note that the bottom section is always set as Flange. The only other option for bottom section is embedded.

In case the Embedded bottom section is selected the program will require the entry of Embed. Depth. The default (and minimum) value of embedment depth is 3 * Bottom Diameter of the bottom section. The embedded joint type means that the pole is fixed in concrete and the program will not perform the base plate calculation.

On the same window a table for ***Section Data*** is displayed. In this table the following data is shown for each section of the pole:

- Section number (from bottom to top)
- Section length (m or ft)
- Section overlap (at top) (m or ft)
- Section bottom diameter (mm or inches)
- Section top diameter (mm or inches)
- Section plate thickness (mm or inches)
- Cross-section type (Circular, 6,8,10,12,16,18,20,24-sides)
- Joint Type (Bottom) (Flange, Telescopic, Embedded)
- Section length lock status (Locked or Unlocked)
- Section yield stress (MPa or ksi)
- Section mass (kg or lbs.)

Based on the total height of the pole and the number of sections, the program calculates equal section lengths with an overlap length equal to 1.5 * diameter at that height. The user may change the section length to match the exact height of the section. Once the section length is changed the new overlap distance, section bottom diameter, top diameter and section mass is calculated

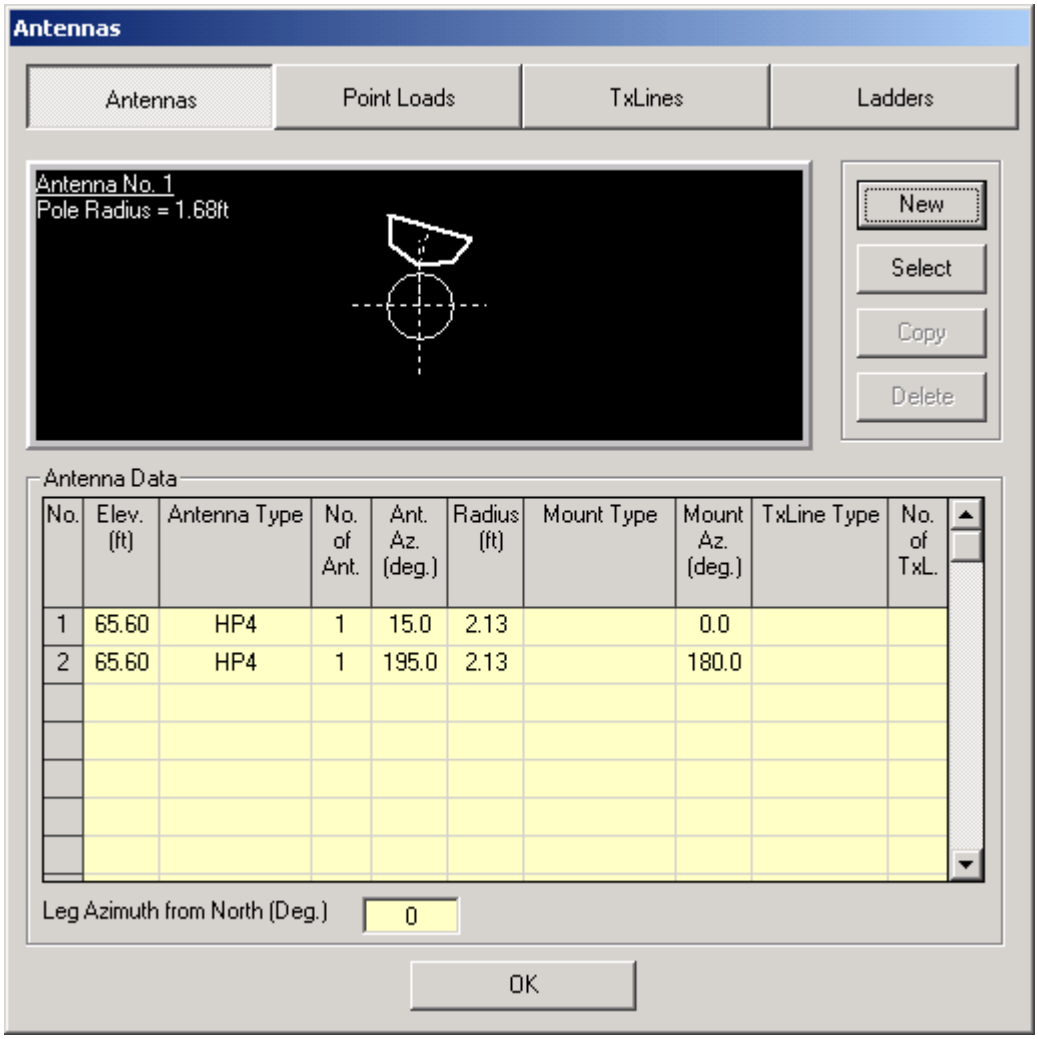
and displayed accordingly and this section becomes locked (i.e. any future change to the total pole height is to be distributed equally to the remaining unlocked sections). At this point the overlap distance may be also changed to any value greater than 1.5 * diameter.

In this table the user may also specify for each section the plate thickness, which may be chosen from the available thickness or typed in the field. The type of cross section may vary from one section to another only if flange type of connection is chosen.

The user may change Joint Type of a given section within this table. The rule imposed on this feature, however, requires that in case of mixed Joint Type sections the Flange Joint sections must be located at the uppermost part of the pole – in other words the program will not permit Telescopic Joint section above Flange Joint section. The exception to this rule, of course, applies to the bottom section, which is always flanged or embedded.

## ANTENNAS DEFINITION

From the main menu choose **Appurtenance** and then select **Antennas,** the following window is displayed.



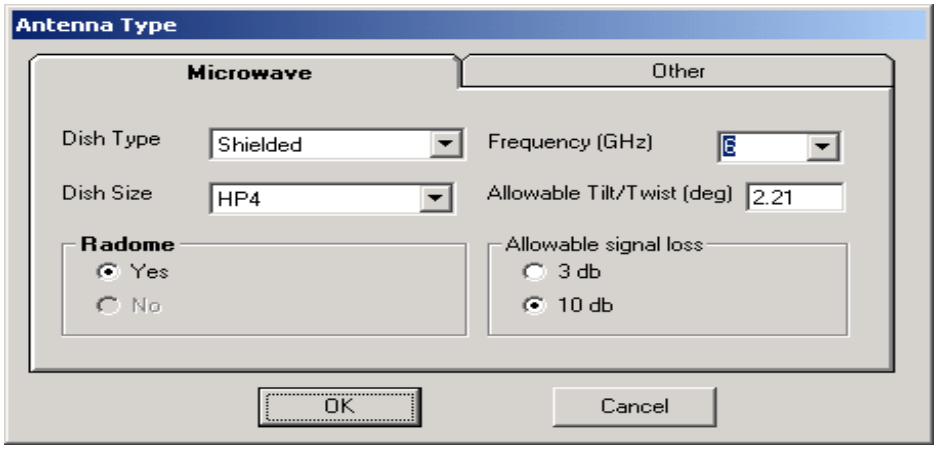

In this window the pole elevation is shown along with the plan cross-section at the marked antenna. To add a new antenna select **New** and a blank line with an antenna type (none) is shown. The user inputs the following data as defined below:

- ***Elevation:*** Elevation of the center of the antenna marked from the bottom of the pole and shown in meters or ft.
- ***Type:*** Type of antenna and can be chosen from the antennas database available. To choose an antenna click on this field and an antenna type window will be displayed and the required type and size is specified.

- ***Azimuth:*** Antenna beam azimuth measured from the zero azimuth of the pole and may be referred to as pole's north (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the mounting point of the antenna (m or ft.). Also, note that the pole radius at that elevation is shown for guidance on the section drawing.
- ***Mounting Azimuth:*** angle between the pole's north and the antenna radius measured in the clockwise direction.

### *Antenna Type:*

Click on type in the antenna table and the following window is displayed and from which the antenna type and size is specified.




For microwave dishes available in the database the following dish types and sizes are available:

| | |
|---|---|
| Shielded | (2, 4, 6, 8, 10, 12, 15 ft.) |
| Focal plane | (4, 6, 8, 10, 12 ft. – with or without radome) |
| Standard | (2, 4, 6, 8, 10, 12 ft. – with or without radome) |
| Grid | (4, 6, 8, 10, 12, 15 ft.) |
| GRIDPAK | (4, 6, 8, 10, 12, 13 ft.) |

Also, for microwave antennas the allowable Tilt/Twist is calculated by the program as a function of frequency, dish diameter and allowable signal degradation (3 db or 10 db) based on the following formula:

a) For a parabolic reflector with an allowable 10dB signal degradation:

$$\theta = \frac{C_{10}}{D \quad \alpha}$$

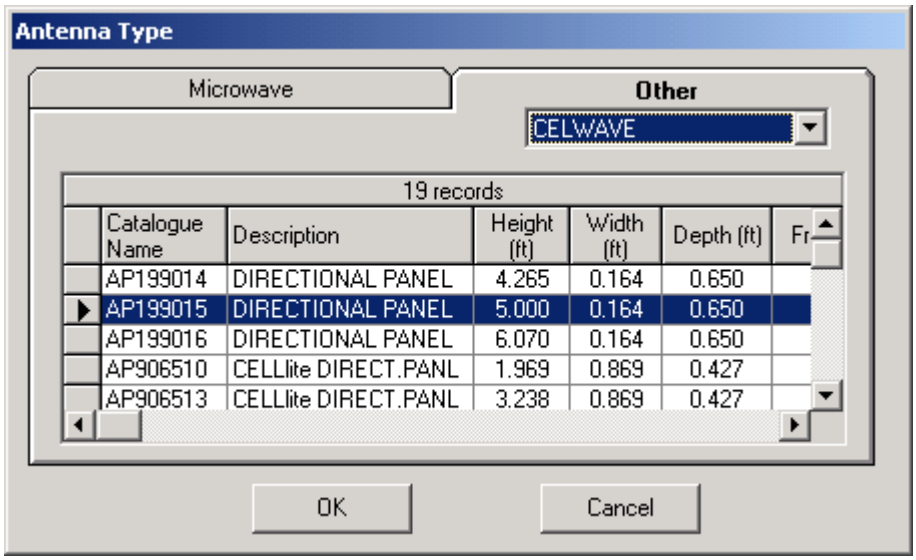
b) For a parabolic reflector with an allowable 3dB signal degradation:

$$\theta = \frac{C_3}{D \quad \alpha}$$

where:

θ = twist or sway limit, degrees
$C_{10}$ = 53.1 GHz.ft.deg [16.2 GHz.m.deg]
$C_3$ = 31.0 GHz.ft.deg [9.45 GHz.m.deg]
D = Diameter of dish, ft [m]
α = Dish Frequency, GHz.

For "Other" (Non-Dish Antennas) a variety of Antennas, organized by manufacturers is available from the database.
Following screen is displayed when the Tab "Other" is clicked.




The data shown includes all parameters of these antennas necessary for the program to calculate wind loads and weight effects.

The User may add any type of Non-Dish Antenna to the database –using the "Database Management" option from the main menu. Refer to Database Management Chapter for details.

After specifying an antenna, the user may wish to copy it by highlighting the specific line and click on **select** and then on **Copy**. A new line of antenna will

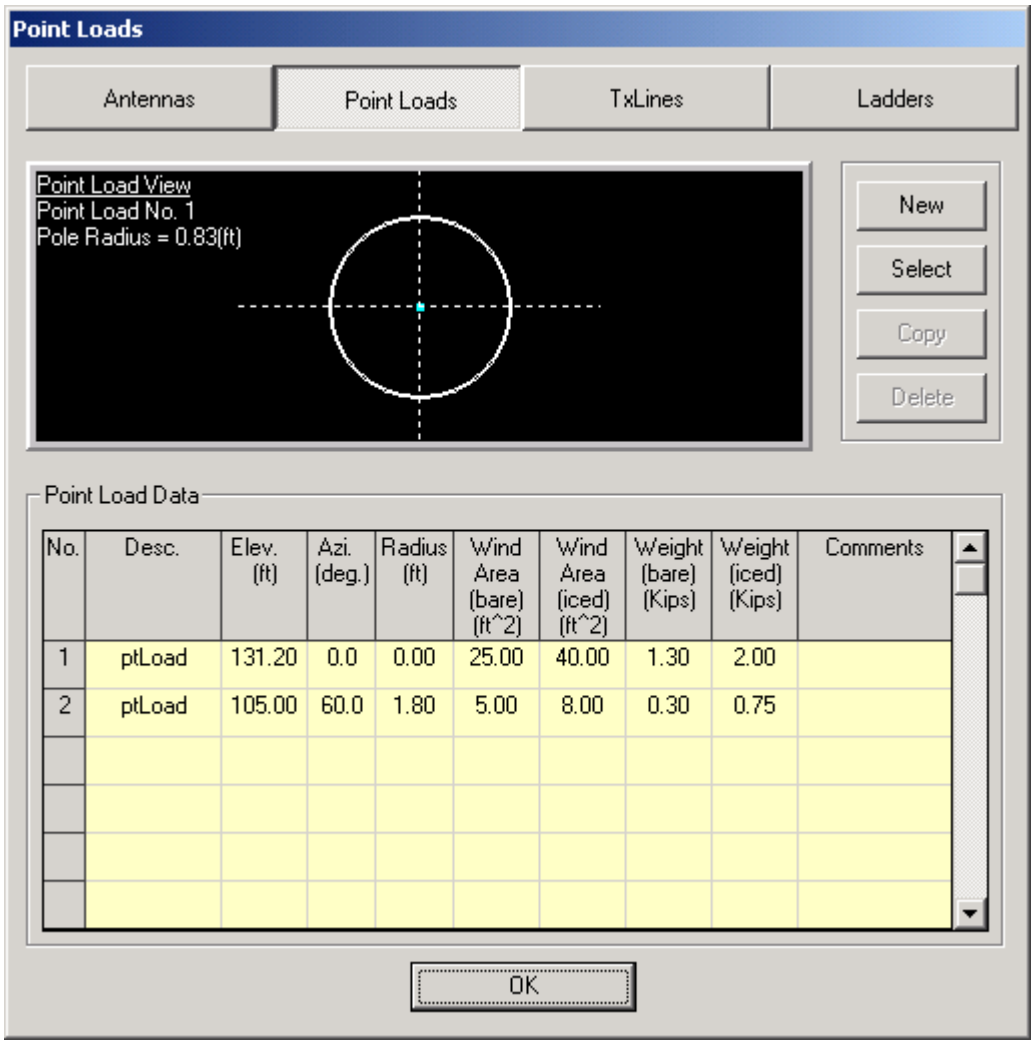
be displayed and the user may edit that line. A similar procedure is used to delete an antenna.

Loads are calculated for microwave dishes as per Andrew's catalogue number 36. Also load calculations for wind loads under different directions are based on EIA-222-F tables.

Also, from this window the user can define the TX-lines type and number that are used for this antenna.

## POINT LOADS DEFINITION

From the main menu choose **Appurtenance** and then select **Point Loads,** the following window is displayed.



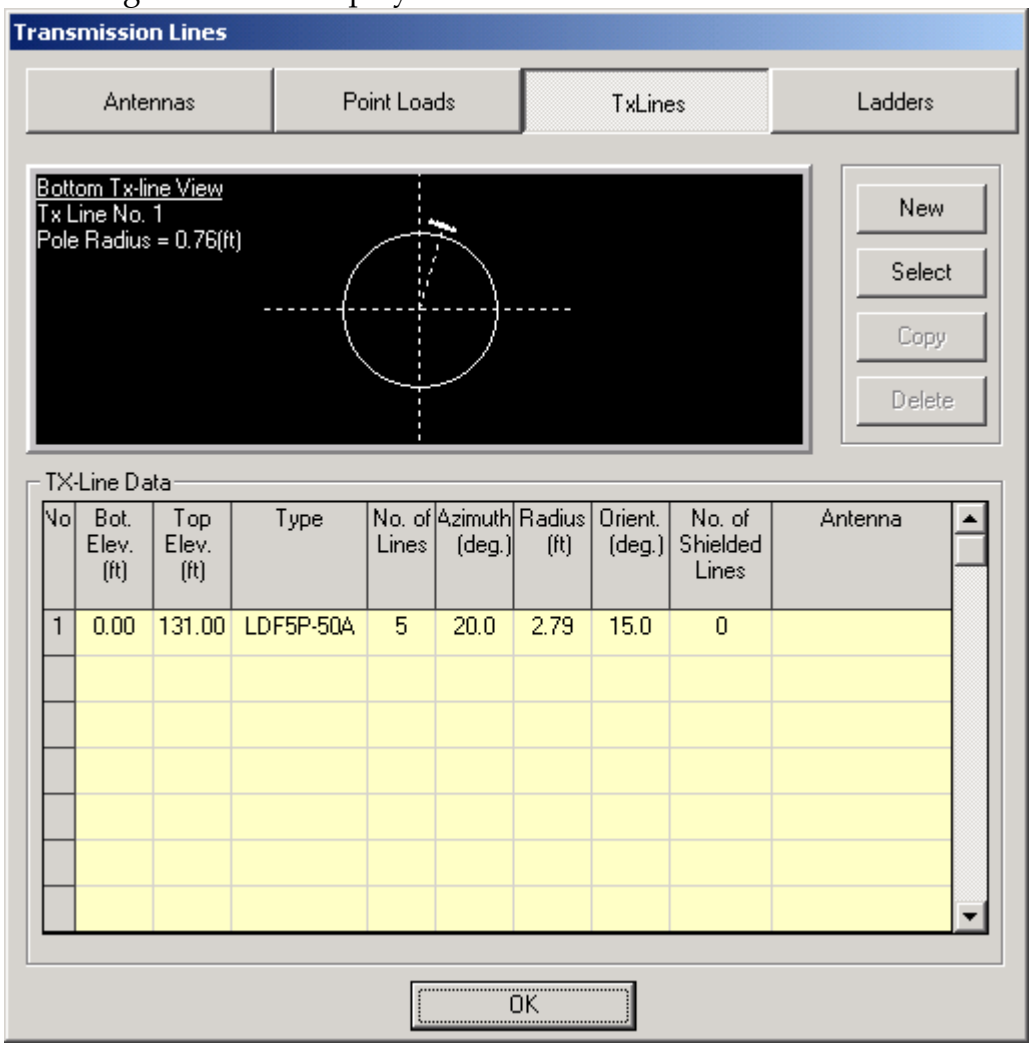

In this window the pole elevation is shown along with the plan cross-section at the point load elevation. To add a new point load select **New** and input line with zero values is shown. The user inputs the following data as defined below:

- ***Description:*** Text description of the point load. This description will be displayed on the design profile.
- ***Elevation:*** Elevation from the bottom of the pole to the center of the applied load (m or ft.)

- ***Azimuth:*** angle between the pole's north and the point load radius measured in the clockwise direction (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the point load (m or ft.). Also, note that the pole radius at that elevation is shown for guidance on the section drawing.
- ***Wind Area (Bare):*** Bare wind area of the load (antenna) multiplied by the appropriate force coefficient or drag factor depending on the shape of the apperuentance ($m^2$ or $ft^2$).
- ***Wind Area (Iced):*** Iced wind area of the load (antenna) multiplied by the appropriate force coefficient or drag factor depending on the shape of the appurentance and the ice accretion ($m^2$ or $ft^2$).
- ***Weight (Bare):*** Bare weight of the load (kN or kips)
- ***Weight (Iced):*** weight of the load including ice (kN or kips)
- ***Comments:*** A comments field that does not get displayed on the profile.

## TRANSMISSION LINES DEFINITION

From the main menu choose **Appurtenance** and then select **Tx-lines,** the following window is displayed.



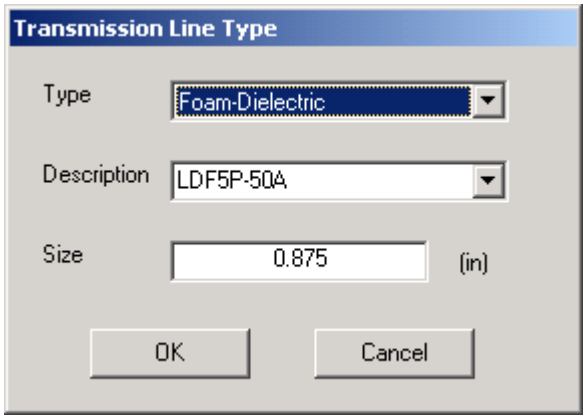

In this window the pole elevation is shown along with the plan cross-section at the bottom of the Tx-line. To add a new line or group of lines select **New** and a blank line with a line type (none) is shown. The user inputs the following data as defined below:

- ***Bottom Elevation:*** Elevation of the bottom of the lines (m or ft.)
- ***Top Elevation:*** Elevation of the top of the lines (m or ft.)
- ***Type:*** Type of lines and can be chosen from the tx-lines database available. To choose line type click on this field and the required type and size can be specified as explained in the sequel.

- ***No of lines.:*** Number of lines having the same properties shown on that line
- ***Azimuth:*** Group of lines azimuth (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the center of the lines group (m or ft.).
- ***Orient.:*** angle between the pole's north and the lines group radius measured in the clockwise direction.
- ***No of Shielded Lines:*** For shielded lines (e.g. inside the pole) only gravity loads are considered.

### *Transmission Line Type:*

Click on type in the tx-lines table and the following window is displayed and from which the tx-lines type and size is specified.




The following Transmission Line types are available:

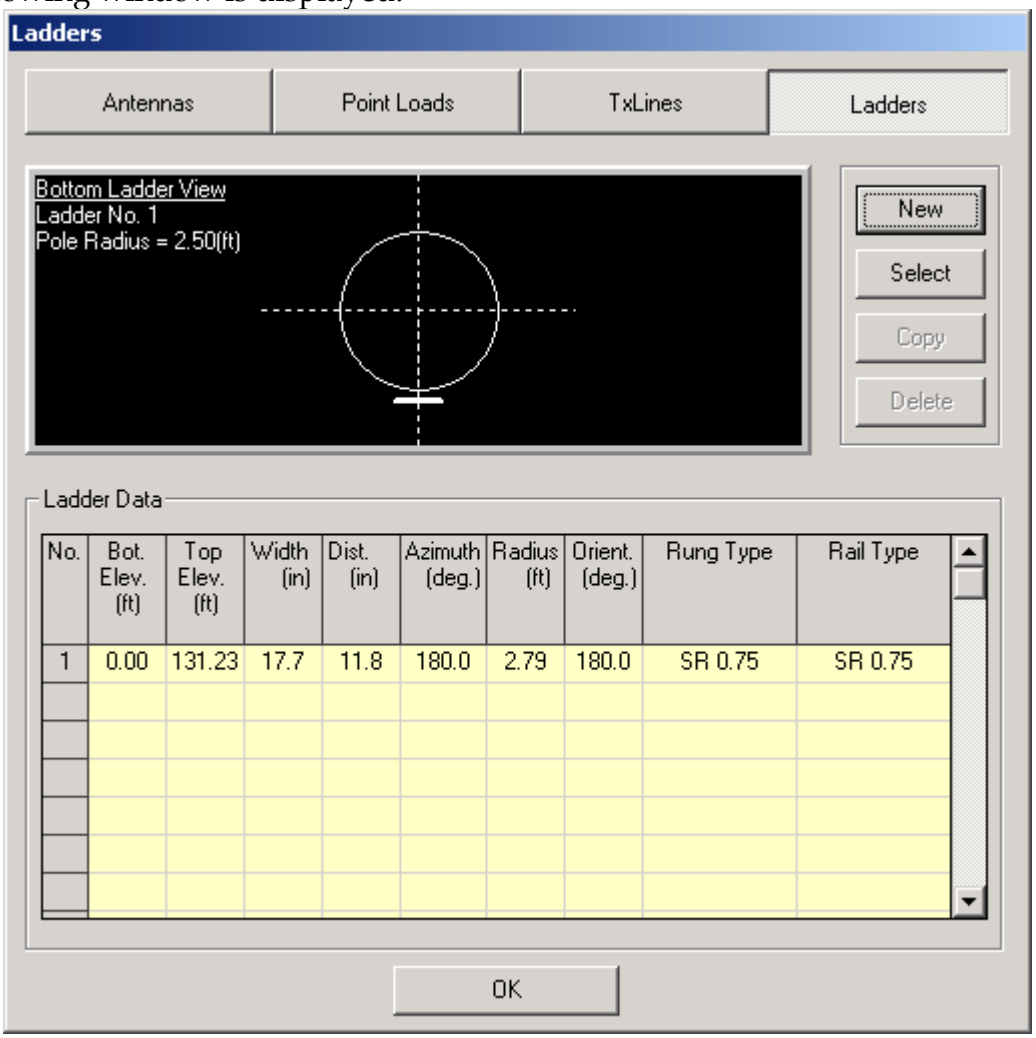
Air-Dielectric
Foam-Dielectric
Elliptical Waveguide
Circular Waveguide
Rectangular Waveguide

Different sizes can be chosen from the description field and the actual size for the chosen lines is displayed in the size field (mm or in.).

Data for transmission lines mechanical properties are based on Andrew's catalogue number 36.

## LADDER DEFINITION

From the main menu choose **Appurtenance** and then select **Ladders,** the following window is displayed.



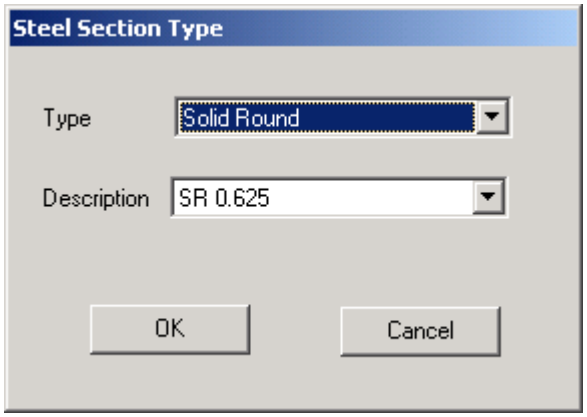

In this window the pole elevation is shown along with the plan cross-section at the bottom of the ladder. To add a new ladder select **New** and a blank line with a rail type, and rung type (none) is shown. The user inputs the following data as defined below:

- ***Bottom Elevation:*** Elevation of the bottom of the ladder (m or ft.)
- ***Top Elevation:*** Elevation of the top of the ladder (m or ft.)
- ***Width:*** Width of ladder rungs (mm or in.)
- ***Dist.:*** Distance (spacing) between rungs (mm or in.)
- ***Azimuth:*** ladder azimuth (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the center of the ladder (m or ft.).

- ***Orient.:*** angle between the pole's north and the ladder's radius measured in the clockwise direction.
- ***Rung Type:*** Rung size chosen from available steel sections database for angles and solid rounds.
- ***Rail Type:*** Rail size chosen from available steel sections database for angles and solid rounds.

### *Steel Section Type:*

Click on rail type or rung type in the ladder data table and the following window is displayed, from which the steel section type and size is specified.




The following steel sections for ladders are available:

Solid Round (SR13, SR16, SR19, SR25, SR50)
(SR.5, SR.625, SR.75, SR1.0, SR2.0)

Angle (L51x51x6, L51x51x13)
(2x2x1/4, l2x2x1/2)

-

# Chapter 3 ANALYSIS

This Chapter explains the input of the code-related data and material data. Also, the assumptions and the underlying theory of the analysis are explained in the sequel.

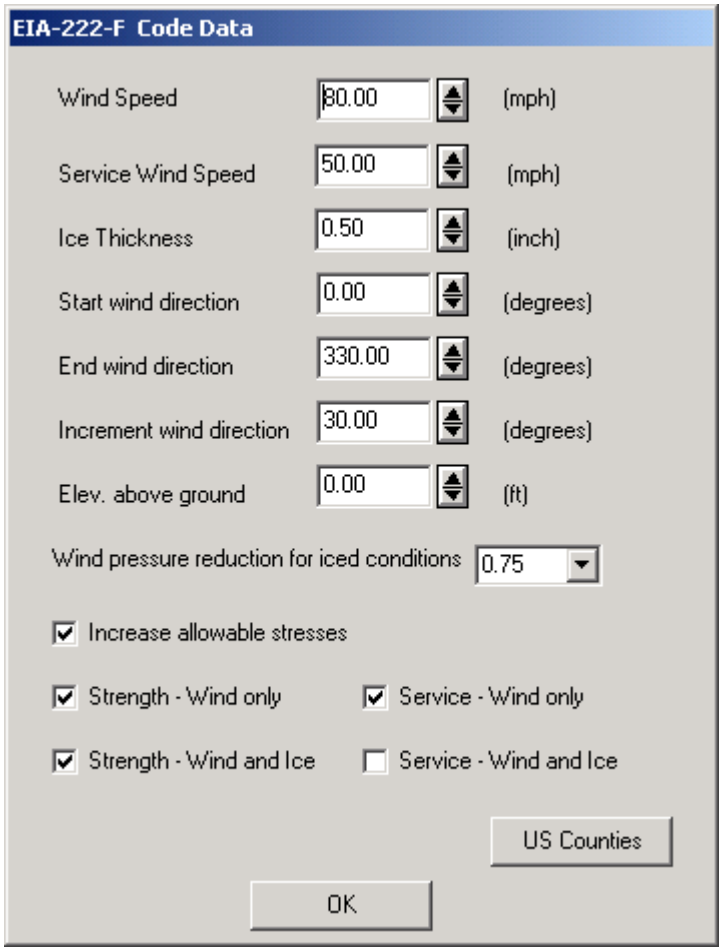
## *CODE DATA*

### Wind Loads

From the main menu choose **Load Cases,** and then he following window will appear depending on the design standard specified:

#### *a) ANSI/TIA/EIA-222-F-1996*

Having the EIA 222-F as the design code, the wind Loads window will be as shown:

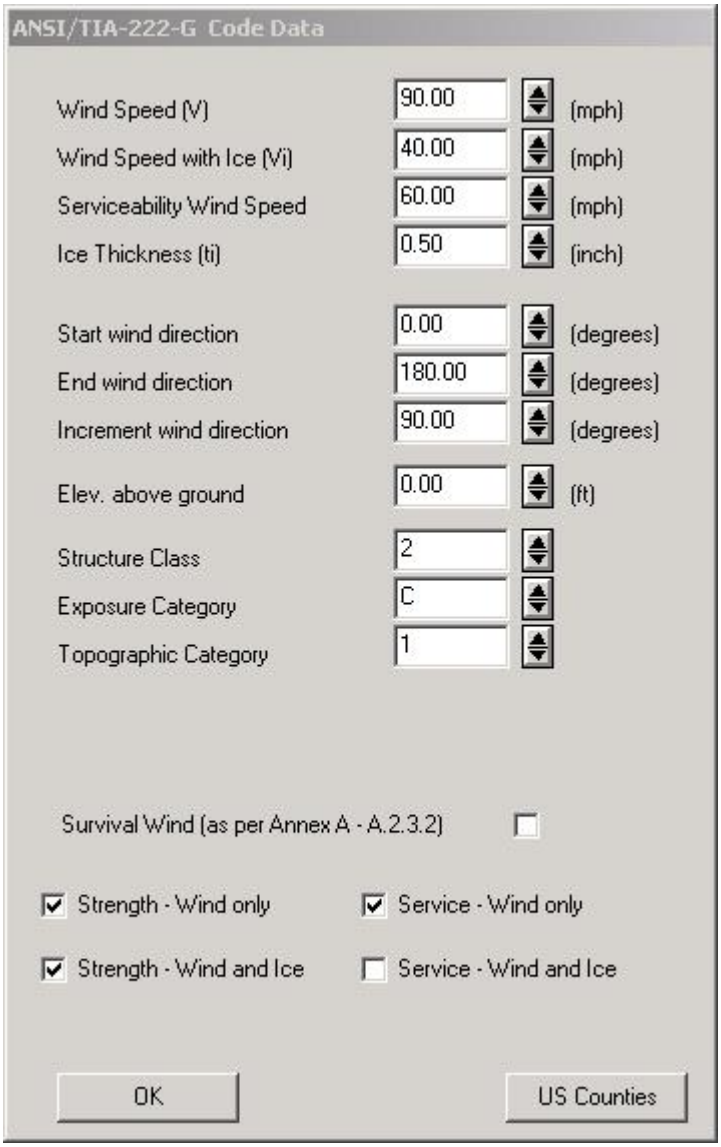


The input data is explained as follows:

- Wind Speed: Design wind speed for the structure (m/sec or mph).
- Service Wind Speed: for the calculation of deflections (tilt/twist) under service load conditions (m/sec or mph).
- Ice Thickness: specified ice thickness for the design of the structure depending on the location (mm or inches).
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered
- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown example wind will be considered at the following directions (0, 30, 60, 330)
- Elevation above ground: The elevation of the structure base above ground this is used for wind loads calculations. The height factor is increased accordingly.
- Allowable stress factor: For the structural assessment of the pole, the allowable stress is increased to 1.33 as per EIA 222-F when applicable. However, the user has the option to limit the allowable stress to 1.0
- Wind pressure reduction for the case of iced conditions (default is 0.75 as per code).
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.

### *b) ANSI/ TIA-222-G-2005*

Selecting the ANSI/TIA-222-G as the design code, the wind Loads window will be displayed as shown:



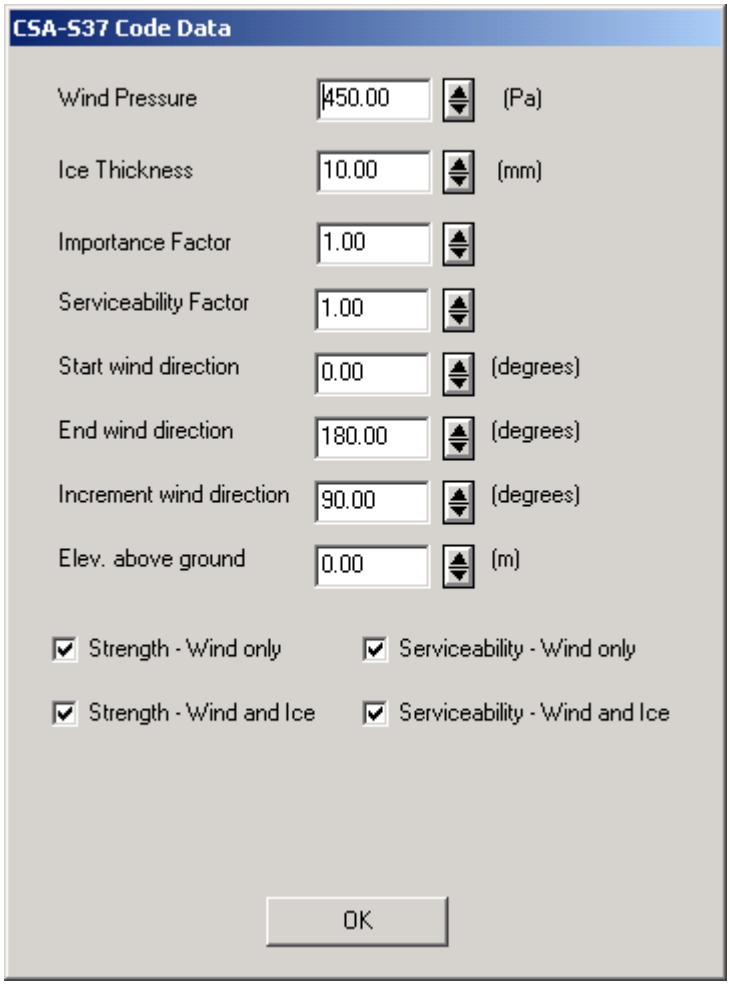

The input data is explained as follows:

- Basic Wind Speed (V): 3-second gust speed for the structure (m/sec or mph).
- Wind Speed with Ice (Vi): 3-second gust speed concurrent with the design ice at 33 ft above the ground (m/sec or mph).
- Serviceability Wind Speed: for the calculation of deflections (tilt/twist) under service load conditions (m/sec or mph).

- Ice Thickness (ti): specified ice thickness for the design of the structure depending on the location (mm or inches).
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered
- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown example wind will be considered at the following directions (0, 90, 180)
- Elevation above ground: The elevation of the structure base above ground this is used for wind loads calculations. The height factor is increased accordingly.
- Structure Class as defined in Table 2-1 of TIA-222-G. Values are 1 through 3 with class 2 as default.
- Exposure Category as defined in clause 2.6.5.1 of TIA-222-G. Options are Exposure B, C or D with Exposure C as default.
- Topographic Category as defined in clause 2.6.6.2 of TIA 222-G ranging from Category 1 through 5 with Category 1 as default.
- Option to use survival wind speed as defined in Annex A of the standards. This option assumes that the wind speed used is a survival wind speed with Load factors, gust factors and height factors of 1.0 as per clause A2.3.2.
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.

### *c) CSA S37-94*

Having the CSA S37-94 as the design code, the wind Loads window will be displayed as shown:



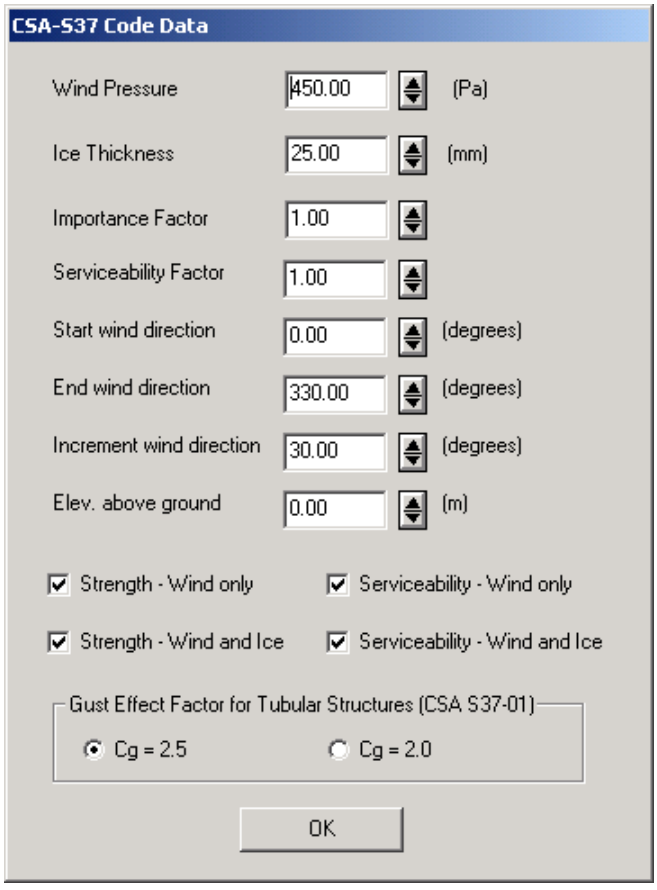

The data for which is explained as follows:

- Wind Pressure: Reference wind pressure (q) as per CSA S37-94
- Ice thickness: Radial ice thickness for the design
- Importance factor: Importance factor as per S37-94
- Serviceability factor: Serviceability factor for service load conditions as per S37-94
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered

- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown window, wind will be considered only from 0 degrees.
- Elevation above ground: The elevation of the structure base above ground. This is used for wind loads calculations.
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.

### *d) CSA S37-01*

Having the CSA S37-01 as the design code, the wind Loads window will be displayed as shown:




In addition to other design parameters explained under S37-94 this window allows the user to choose the value of $C_g$ as per CSA S37-01. If $C_g$ = 2.0 is selected, then the program will check if the condition 4.6.2 (a) of S37-01 is satisfied. If that is the case, then no other checkups are performed and the program applies $C_g$ = 2.0.

If, however, the condition in clause 4.6.2 (a) is not satisfied then the program uses the clause 4.6.2 (b) and adds one more loading case (Vortex Shedding) to the analysis. The calculation of the applied loads in this case is based on the Supplement to the National Building Code of Canada 1990, Commentary B, Wind Loads. An important parameter for these calculations is the critical damping ratio β. The Engineer should enter or verify the value of the critical damping ratio (accessed through the "Calculation Parameters" sub-menu of the Setup Menu. The default value is β = 0.004.

## *ANALYSIS THEORY*

The analysis is based on three-dimensional beam model with geometrical non-linear capabilities. Increased moment due to P-delta effects are accounted for in the analysis. Wind load calculations and analysis are based on the following assumptions:

### Wind Loads Calculations

1. Wind is assumed horizontal and is blowing from a certain direction throughout the whole pole height.

2. Force coefficients (drag factors) are calculated as per applicable code.

3. Load calculations for multi-sided poles not explicitly spelled out in the codes (poles with number of sides greater than 16) are based on linear interpolation between 16-sided poles and circular poles.

4. No shielding is considered on the pole from antennas, Tx-lines, or ladders.

5. Tx-lines and ladders located inside the pole are considered shielded by the pole.

6. Ice built up is considered uniform on the structure, external tx-lines and ladders.

7. Wind loads considered on iced structures are reduced as per applicable code.

8. For wind load calculations, each pole section is discretized into 5 sub-sections.

9. Loads that are offset from the tower center are applied at the tower center with the corresponding moments (torsional and bending).

10. Loads that extend beyond the height of the structure are applied at the top of the structure with the corresponding additional moments (torsional and bending).

### Analysis and Capacities

1. Each pole section is modeled as five elements.

2. Each element is modeled as two-nodded three-dimensional beam elements with six degrees of freedom at each node.

3. Element properties are calculated based on mid-height dimensions.

4. Pole is considered fixed in all directions at the base.

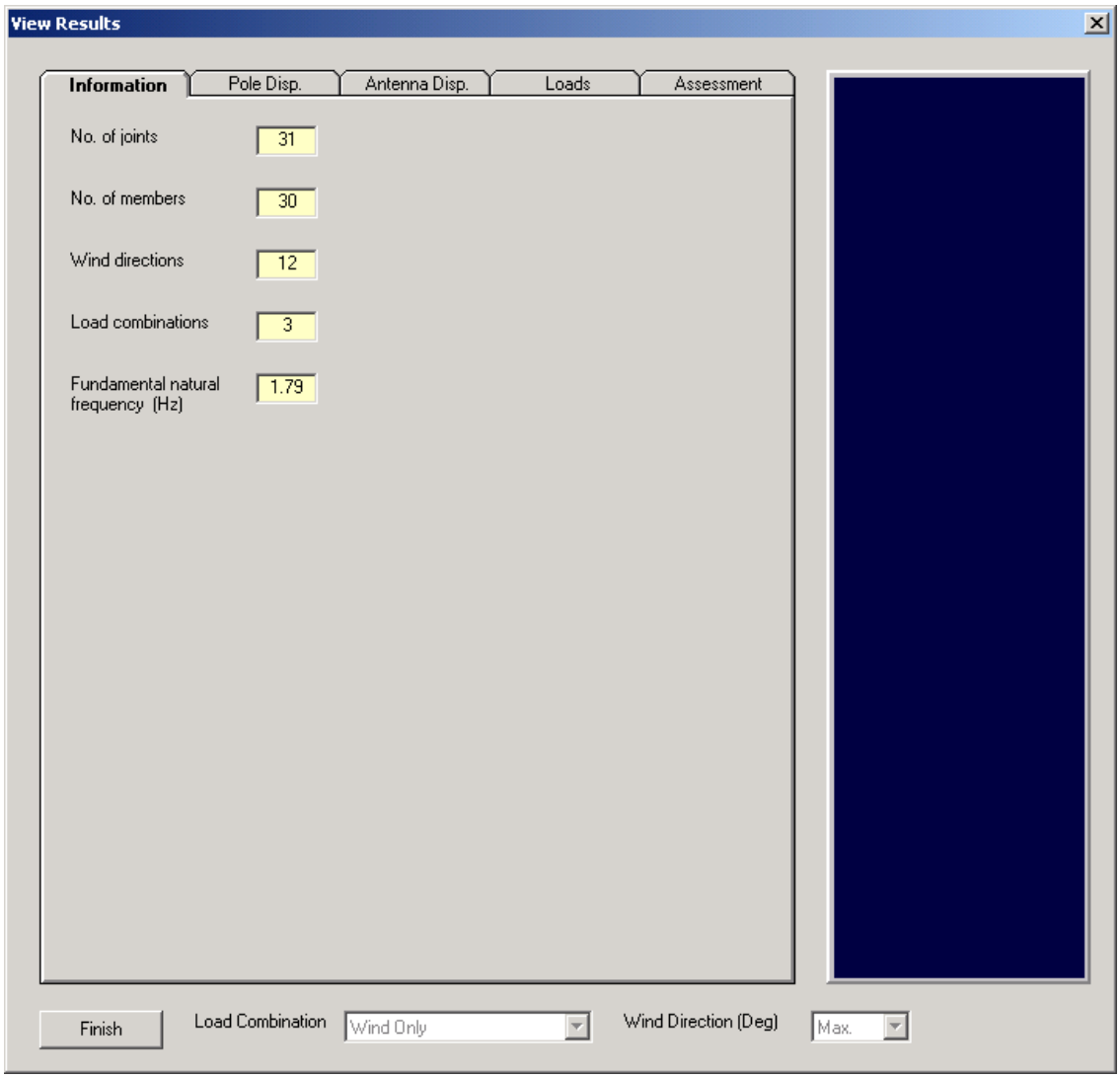
5. Uniform loads applied to the element are distributed to the top and bottom element nodes.

6. Capacities are calculated based on applicable codes and the structure is assessed at each element.

7. Capacities for 8, and 10 sided poles are based on 8 sided equations. Poles with number of sides greater than 16 sides are based on 16 sided equations.

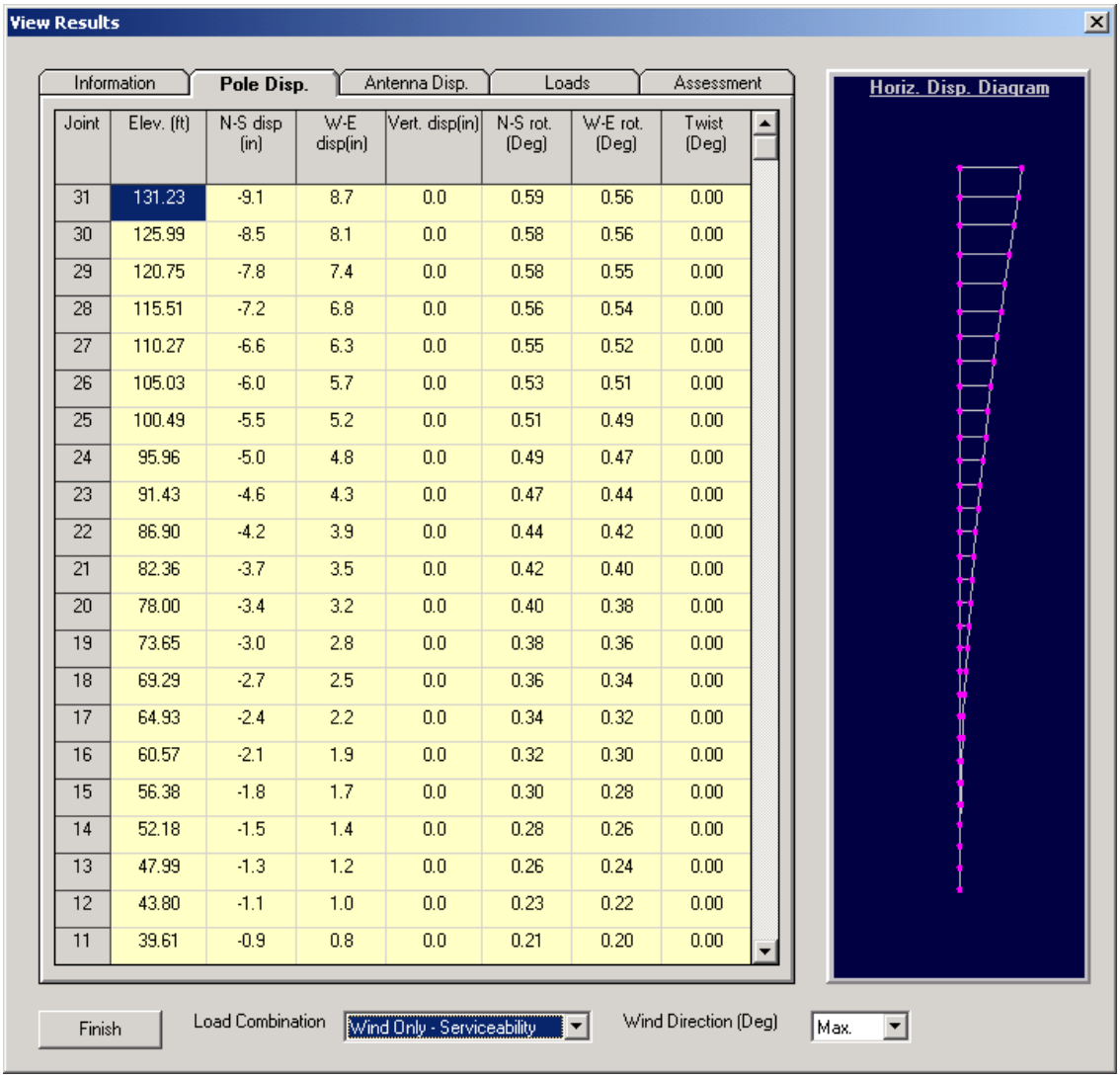
# Chapter 4 OUTPUT

In this Chapter the printed and viewable output options for the program are explained. After the input data phase is completed, the user chooses **Run** from the main menu. The program performs the wind load calculations and executes the structural analysis. From the main menu the user may choose **Results** and the following screen is displayed which shows the number of joints (nodes), number of members (elements), number of wind directions considered for each load combination, and number of load combinations.



## Pole Displacement

The user may view the pole displacement on screen by choosing the **Pole Disp.** tab from the **Results** menu.



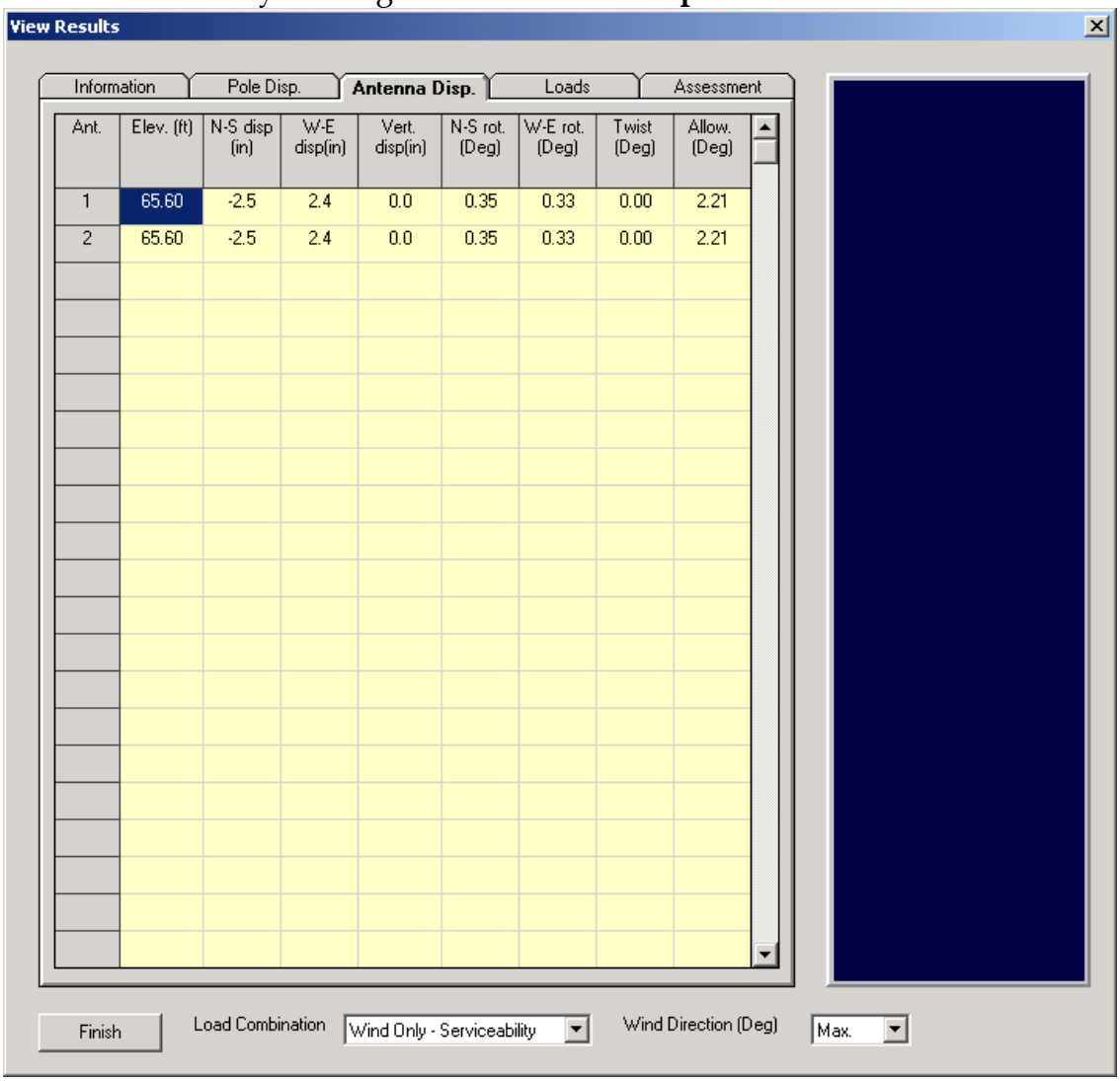

| Joint | Elev. (ft) | N-S disp (in) | W-E disp(in) | Vert. disp(in) | N-S rot. (Deg) | W-E rot. (Deg) | Twist (Deg) |
|---|---|---|---|---|---|---|---|
| 31 | 131.23 | -9.1 | 8.7 | 0.0 | 0.59 | 0.56 | 0.00 |
| 30 | 125.99 | -8.5 | 8.1 | 0.0 | 0.58 | 0.56 | 0.00 |
| 29 | 120.75 | -7.8 | 7.4 | 0.0 | 0.58 | 0.55 | 0.00 |
| 28 | 115.51 | -7.2 | 6.8 | 0.0 | 0.56 | 0.54 | 0.00 |
| 27 | 110.27 | -6.6 | 6.3 | 0.0 | 0.55 | 0.52 | 0.00 |
| 26 | 105.03 | -6.0 | 5.7 | 0.0 | 0.53 | 0.51 | 0.00 |
| 25 | 100.49 | -5.5 | 5.2 | 0.0 | 0.51 | 0.49 | 0.00 |
| 24 | 95.96 | -5.0 | 4.8 | 0.0 | 0.49 | 0.47 | 0.00 |
| 23 | 91.43 | -4.6 | 4.3 | 0.0 | 0.47 | 0.44 | 0.00 |
| 22 | 86.90 | -4.2 | 3.9 | 0.0 | 0.44 | 0.42 | 0.00 |
| 21 | 82.36 | -3.7 | 3.5 | 0.0 | 0.42 | 0.40 | 0.00 |
| 20 | 78.00 | -3.4 | 3.2 | 0.0 | 0.40 | 0.38 | 0.00 |
| 19 | 73.65 | -3.0 | 2.8 | 0.0 | 0.38 | 0.36 | 0.00 |
| 18 | 69.29 | -2.7 | 2.5 | 0.0 | 0.36 | 0.34 | 0.00 |
| 17 | 64.93 | -2.4 | 2.2 | 0.0 | 0.34 | 0.32 | 0.00 |
| 16 | 60.57 | -2.1 | 1.9 | 0.0 | 0.32 | 0.30 | 0.00 |
| 15 | 56.38 | -1.8 | 1.7 | 0.0 | 0.30 | 0.28 | 0.00 |
| 14 | 52.18 | -1.5 | 1.4 | 0.0 | 0.28 | 0.26 | 0.00 |
| 13 | 47.99 | -1.3 | 1.2 | 0.0 | 0.26 | 0.24 | 0.00 |
| 12 | 43.80 | -1.1 | 1.0 | 0.0 | 0.23 | 0.22 | 0.00 |
| 11 | 39.61 | -0.9 | 0.8 | 0.0 | 0.21 | 0.20 | 0.00 |

On this screen, the user can see the joint number (for this example, the tower has 3 sections and 5 elements per section, 16 joints), and the elevation at each joint number, displacements in the orthogonal three directions, rotations and twist.

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

## Antenna Displacement

The user may view the antenna displacement at the location of microwave antennas chosen by clicking on the **Antenna Disp.** tab from the **Results** menu.

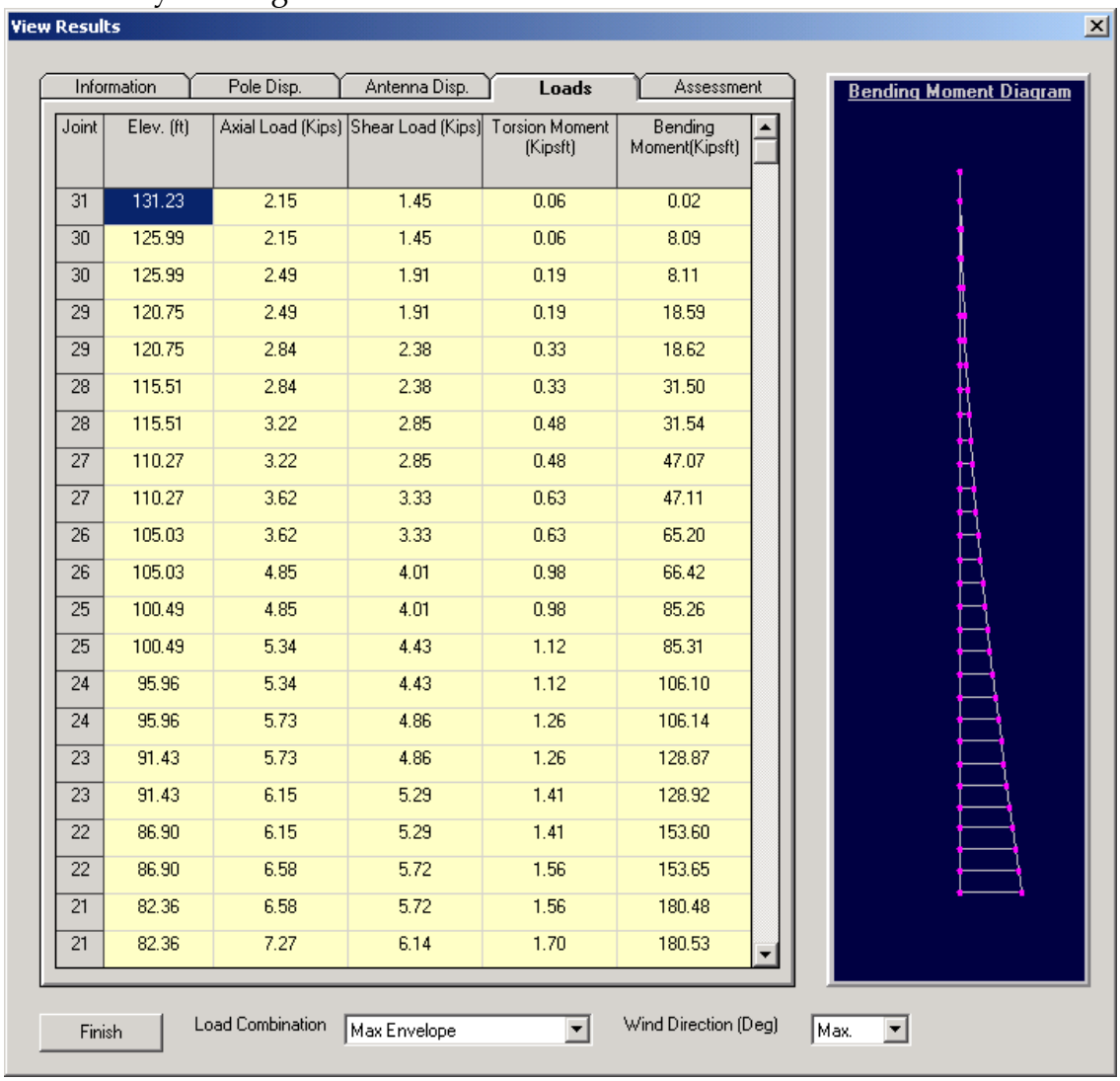



On this screen, the user can see the antenna number (for this example, the tower has 2 antenna at 20m elevation), and the elevation at each antenna, displacements in the orthogonal three directions, rotations and twist.

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

## Loads Output

The user may view the output loads (forces and moments) at each of the elements by clicking on the **Loads** tab from the **Results** menu.



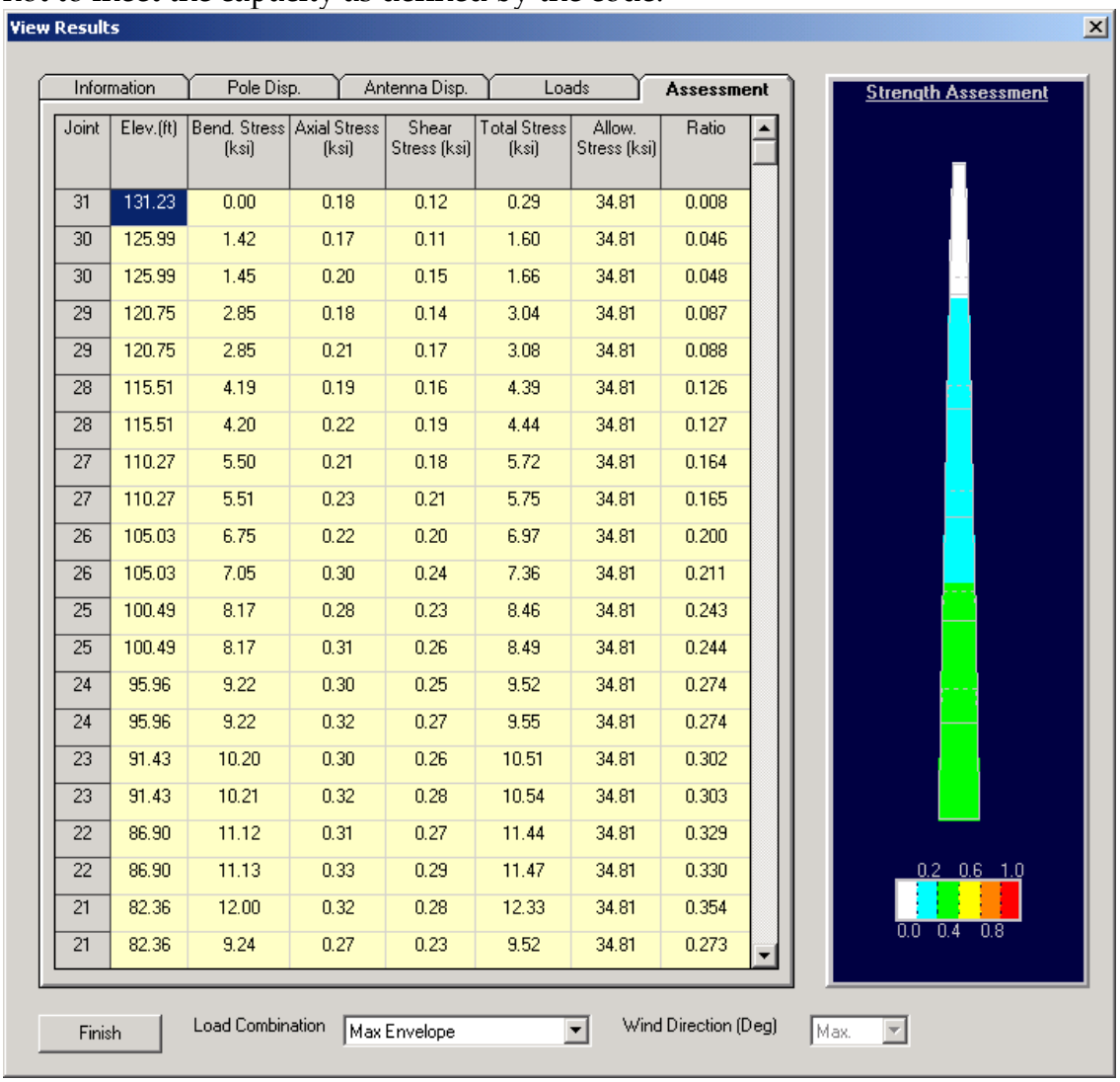

| Joint | Elev. (ft) | Axial Load (Kips) | Shear Load (Kips) | Torsion Moment (Kipsft) | Bending Moment(Kipsft) |
|---|---|---|---|---|---|
| 31 | 131.23 | 2.15 | 1.45 | 0.06 | 0.02 |
| 30 | 125.99 | 2.15 | 1.45 | 0.06 | 8.09 |
| 30 | 125.99 | 2.49 | 1.91 | 0.19 | 8.11 |
| 29 | 120.75 | 2.49 | 1.91 | 0.19 | 18.59 |
| 29 | 120.75 | 2.84 | 2.38 | 0.33 | 18.62 |
| 28 | 115.51 | 2.84 | 2.38 | 0.33 | 31.50 |
| 28 | 115.51 | 3.22 | 2.85 | 0.48 | 31.54 |
| 27 | 110.27 | 3.22 | 2.85 | 0.48 | 47.07 |
| 27 | 110.27 | 3.62 | 3.33 | 0.63 | 47.11 |
| 26 | 105.03 | 3.62 | 3.33 | 0.63 | 65.20 |
| 26 | 105.03 | 4.85 | 4.01 | 0.98 | 66.42 |
| 25 | 100.49 | 4.85 | 4.01 | 0.98 | 85.26 |
| 25 | 100.49 | 5.34 | 4.43 | 1.12 | 85.31 |
| 24 | 95.96 | 5.34 | 4.43 | 1.12 | 106.10 |
| 24 | 95.96 | 5.73 | 4.86 | 1.26 | 106.14 |
| 23 | 91.43 | 5.73 | 4.86 | 1.26 | 128.87 |
| 23 | 91.43 | 6.15 | 5.29 | 1.41 | 128.92 |
| 22 | 86.90 | 6.15 | 5.29 | 1.41 | 153.60 |
| 22 | 86.90 | 6.58 | 5.72 | 1.56 | 153.65 |
| 21 | 82.36 | 6.58 | 5.72 | 1.56 | 180.48 |
| 21 | 82.36 | 7.27 | 6.14 | 1.70 | 180.53 |

On this screen, the user can see the element number, elevation, shear forces in both directions, bending moment in the two orthogonal directions and the twisting moment.

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

### Structural Assessment

The program performs a structural assessment based on the applicable code. The results are displayed in the format of contour lines based on the usage factor. In general, a usage factor greater than 1.0 (shown in red) is considered not to meet the capacity as defined by the code.




| Joint | Elev.(ft) | Bend. Stress (ksi) | Axial Stress (ksi) | Shear Stress (ksi) | Total Stress (ksi) | Allow. Stress (ksi) | Ratio |
|---|---|---|---|---|---|---|---|
| 31 | 131.23 | 0.00 | 0.18 | 0.12 | 0.29 | 34.81 | 0.008 |
| 30 | 125.99 | 1.42 | 0.17 | 0.11 | 1.60 | 34.81 | 0.046 |
| 30 | 125.99 | 1.45 | 0.20 | 0.15 | 1.66 | 34.81 | 0.048 |
| 29 | 120.75 | 2.85 | 0.18 | 0.14 | 3.04 | 34.81 | 0.087 |
| 29 | 120.75 | 2.85 | 0.21 | 0.17 | 3.08 | 34.81 | 0.088 |
| 28 | 115.51 | 4.19 | 0.19 | 0.16 | 4.39 | 34.81 | 0.126 |
| 28 | 115.51 | 4.20 | 0.22 | 0.19 | 4.44 | 34.81 | 0.127 |
| 27 | 110.27 | 5.50 | 0.21 | 0.18 | 5.72 | 34.81 | 0.164 |
| 27 | 110.27 | 5.51 | 0.23 | 0.21 | 5.75 | 34.81 | 0.165 |
| 26 | 105.03 | 6.75 | 0.22 | 0.20 | 6.97 | 34.81 | 0.200 |
| 26 | 105.03 | 7.05 | 0.30 | 0.24 | 7.36 | 34.81 | 0.211 |
| 25 | 100.49 | 8.17 | 0.28 | 0.23 | 8.46 | 34.81 | 0.243 |
| 25 | 100.49 | 8.17 | 0.31 | 0.26 | 8.49 | 34.81 | 0.244 |
| 24 | 95.96 | 9.22 | 0.30 | 0.25 | 9.52 | 34.81 | 0.274 |
| 24 | 95.96 | 9.22 | 0.32 | 0.27 | 9.55 | 34.81 | 0.274 |
| 23 | 91.43 | 10.20 | 0.30 | 0.26 | 10.51 | 34.81 | 0.302 |
| 23 | 91.43 | 10.21 | 0.32 | 0.28 | 10.54 | 34.81 | 0.303 |
| 22 | 86.90 | 11.12 | 0.31 | 0.27 | 11.44 | 34.81 | 0.329 |
| 22 | 86.90 | 11.13 | 0.33 | 0.29 | 11.47 | 34.81 | 0.330 |
| 21 | 82.36 | 12.00 | 0.32 | 0.28 | 12.33 | 34.81 | 0.354 |
| 21 | 82.36 | 9.24 | 0.27 | 0.23 | 9.52 | 34.81 | 0.273 |

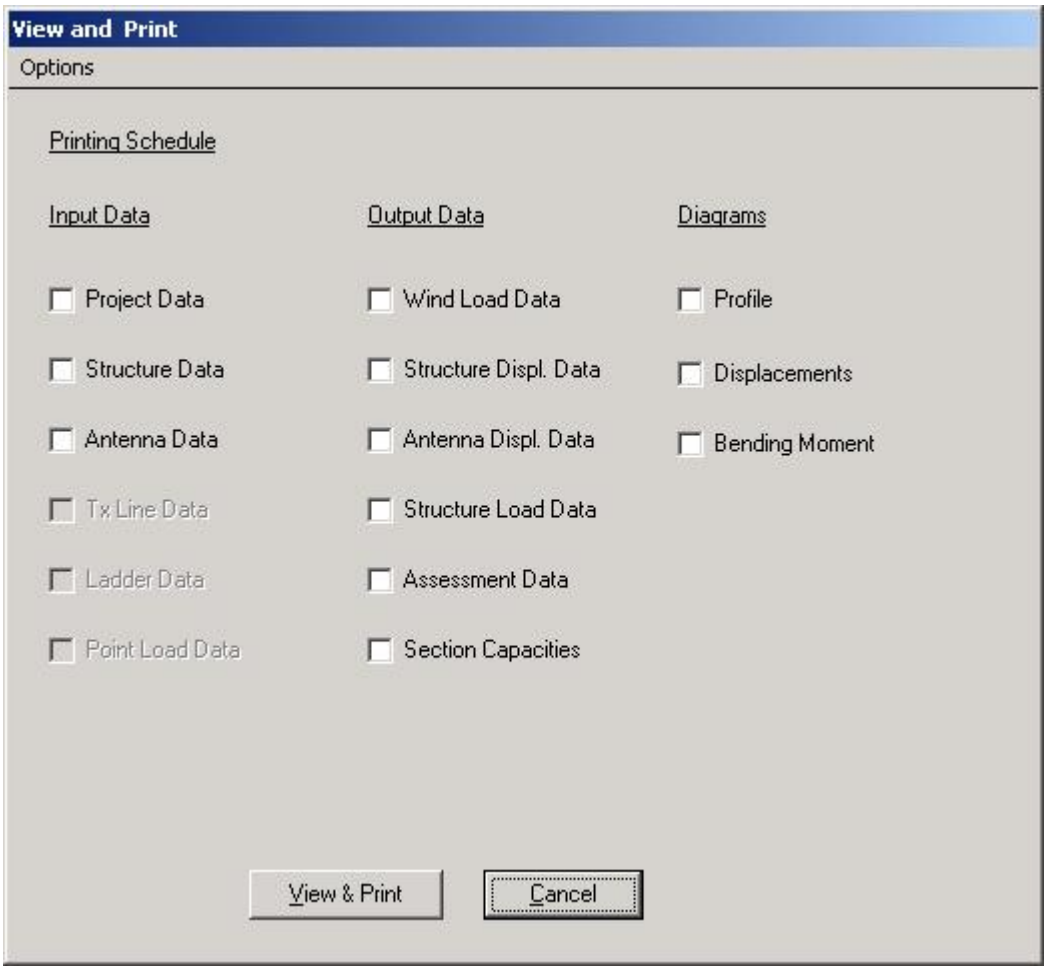
On this screen, the user can see the element number, elevation, bending stresses, shear stresses, total stresses, allowable stress, and the usage ratio defined as the ratio between the actual maximum stress and the allowable stress at this elevation.

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. This screen is calculated only for

the maximum loads of the specific load combination regardless of the wind direction.

### Printout

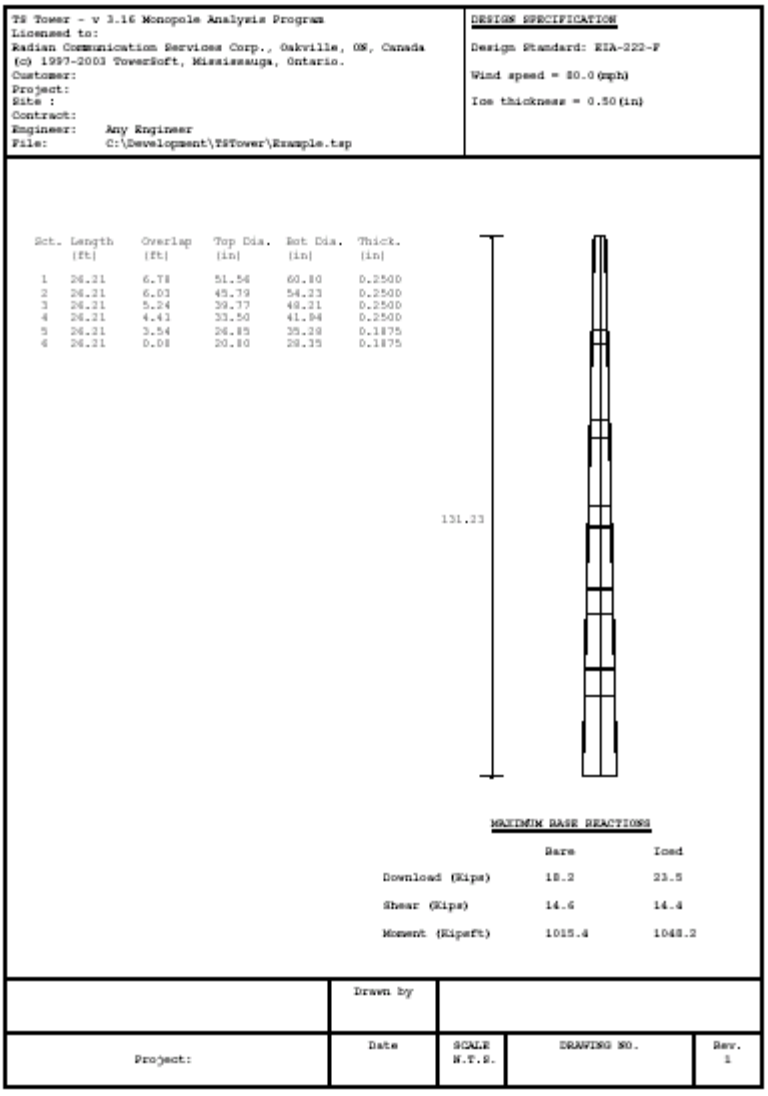
From the main menu, click on **View and Print Results**, and the following screen will be displayed. From this screen the user may choose the required printout sections by clicking on the corresponding check box.




Note that for analysis including several load combinations and different wind directions as well the user has the option to select the required sections for printing and also the desired wind directions. . This may result in a relatively large number of pages in the printout.

The diagram option allows the user to print a simple profile on which the project data, design specification, base reactions and pole geometry are printed in a graphical format. The profile for the example tower is shown in the next figure.

The resulting print out is a Rich Text File (.rtf) that can be viewed and printed by either MS Word or Word Pad. The header of the output file will show the Program information, file information as well as the licensee's information.



TS Tower - v 3.16 Monopole Analysis Program
Licensed to:
Radian Communication Services Corp., Oakville, ON, Canada
(c) 1997-2003 TowerSoft, Mississauga, Ontario.
Customer:
Project:
Site :
Contract:
Engineer: Any Engineer
File: C:\Development\TSTower\Example.tsp

DESIGN SPECIFICATION

Design Standard: EIA-222-F

Wind speed = 80.0 (mph)

Ice thickness = 0.50 (in)

| Sct. | Length (ft) | Overlap (ft) | Top Dia. (in) | Bot Dia. (in) | Thick. (in) |
|---|---|---|---|---|---|
| 1 | 26.21 | 6.78 | 51.56 | 60.80 | 0.2500 |
| 2 | 26.21 | 6.03 | 45.79 | 54.23 | 0.2500 |
| 3 | 26.21 | 5.24 | 39.77 | 48.21 | 0.2500 |
| 4 | 26.21 | 4.41 | 33.50 | 41.94 | 0.2500 |
| 5 | 26.21 | 3.54 | 26.85 | 35.28 | 0.1875 |
| 6 | 26.21 | 0.00 | 20.80 | 28.35 | 0.1875 |

131.33

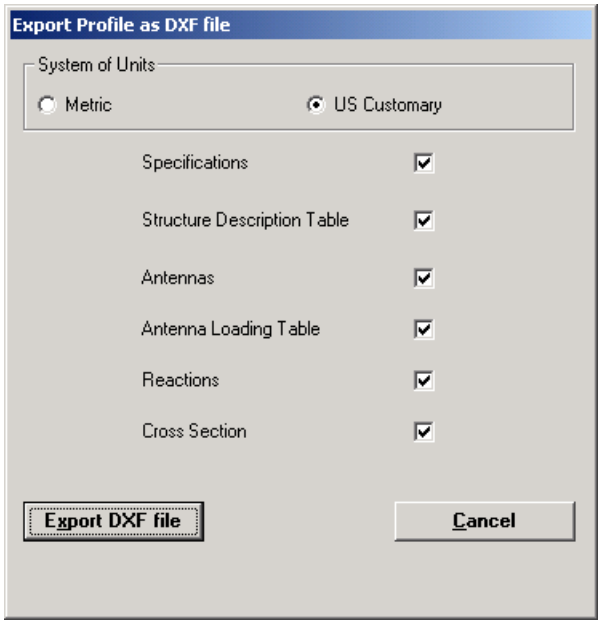
MAXIMUM BASE REACTIONS

| | Bare | Iced |
|---|---|---|
| Download (Kips) | 18.2 | 23.5 |
| Shear (Kips) | 14.6 | 14.4 |
| Moment (Kipsft) | 1015.4 | 1048.2 |

| | Drawn by | | |
|---|---|---|---|
| Project: | Date | SCALE N.T.S. | DRAWING NO. | Rev. 1 |

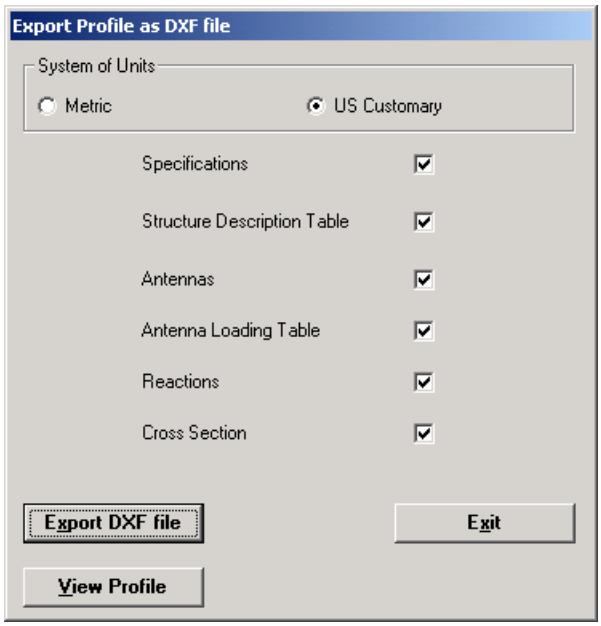
## Export Profile

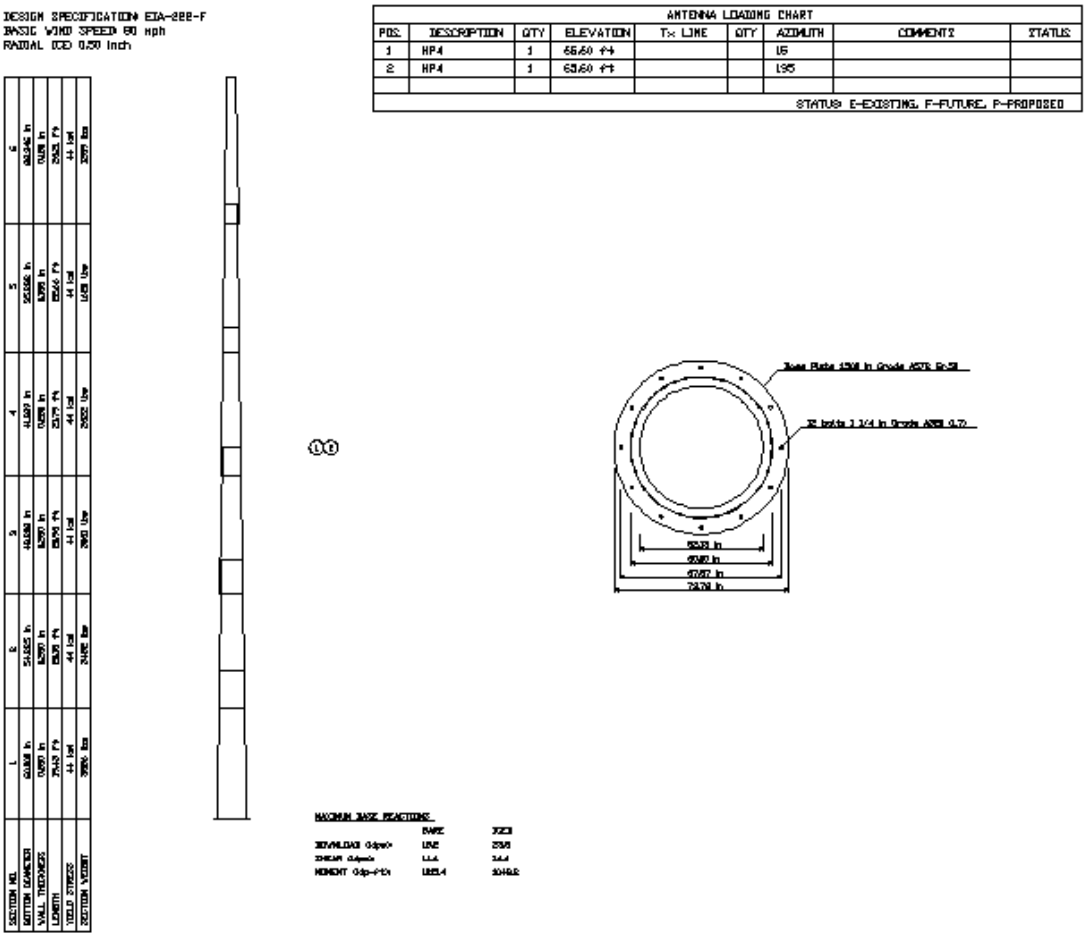
Select sub-menu "**Export Profile (*.dxf)**" from main menu "**Results**". Following screen will be displayed.




Click on "**Export DXF file**" button.

If you have selected DXF viewer you will find additional button "**View Profile**" as shown below.

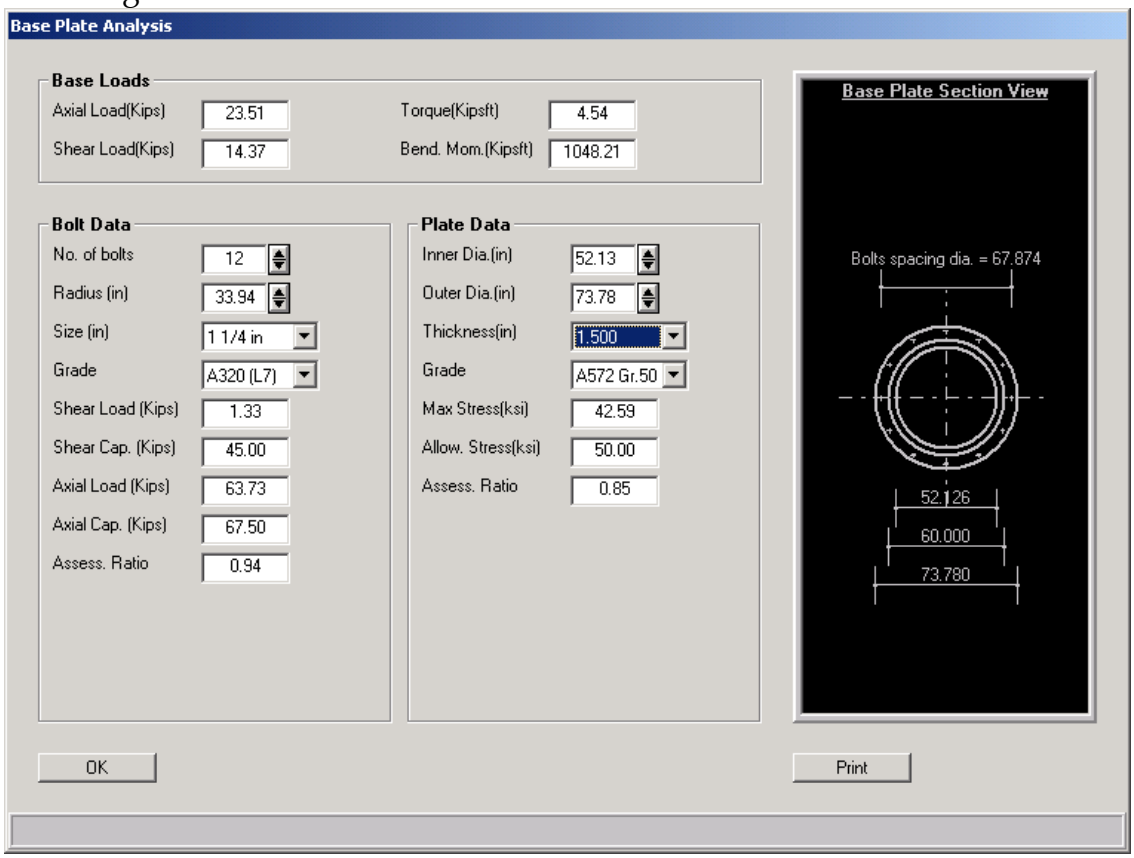





Attached is printout of the dxf profile from the example.

# Chapter 5 BASE PLATE

After the analysis is complete, the user may check the design of the base plate including the anchor bolts.




From the main menu, choose ***Connections*** and then ***Base Plate*** and the above window is displayed. The reactions from the analysis are displayed at the top portion of the screen and the screen is divided into two frames one for bolts and the other for plate data.

The bolt data is calculated based on the following:

- Tension and compression resulting from base moments is resisted by bolts and calculated based on the bolt diameter circle
- Shear is distributed equally to the number of bolts
- The user may change bolt radius, size of bolts and number of bolts.
- The shear load, shear capacity, axial load and axial capacity are calculated and shown on the bolt data frame.

- Bolt assessment is shown based on the combined axial and shear stresses in the bolts and printed on the bolt data frame

## Base Plate

On the base plate window, the user may change the inner base plate diameter (which is only shown for the output diagram purpose), the outer diameter, plate thickness, and steel grade.

Based on the number of bolts the program calculates the influence area for each bolt and the moment is calculated at the intersection between the base plate and the pole. The bending moment is calculated based on the cantilever distance from the center of the bolt to the pole edge. Stresses are calculated based on the plate thickness and the tributary perimeter of the plate. The plate is assessed as per applicable code and result is shown in the assessment ratio.

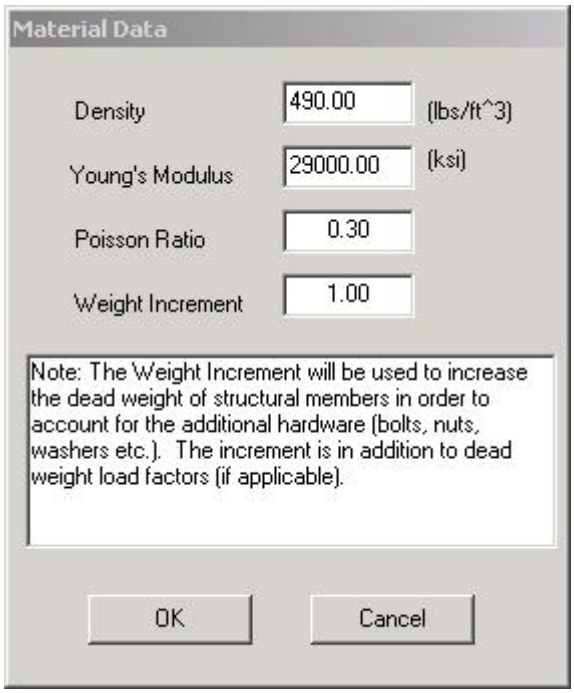
# Chapter 6 SETUP

The program uses defaults based on the selections by user selected options below:

## Material Data

Clicking on **Setup** from the main menu the material data can be defined. The user inputs the material density (lbs/ft$^3$ or kg/m$^3$), Young's modulus (ksi or MPa) and Poisson's ratio.

Also, a weight increment (factor) which can be used to increase dead weight of structural members to account for connecting hardware (bolts, nuts, etc.), galvanizing, etc.

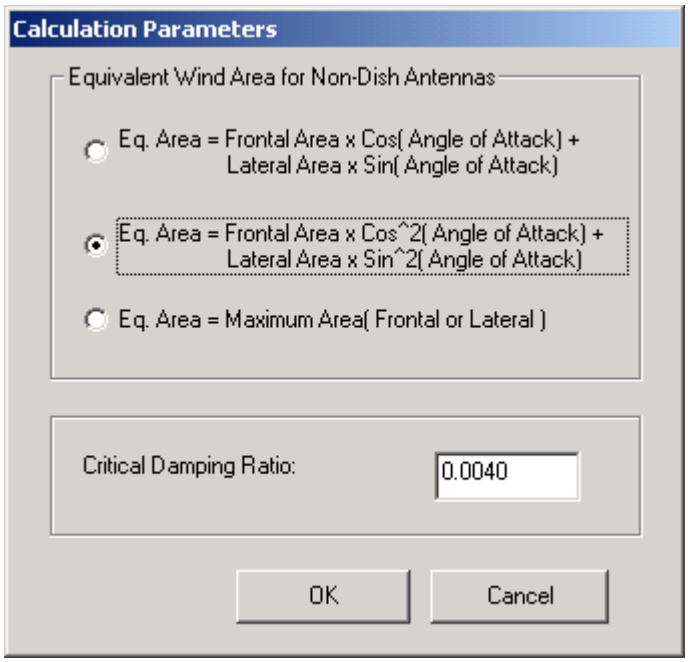



## Default Standards, Units & Analysis Engine

In this menu, the user may choose to select the default standards used for the analysis and select also the default units for the analysis, the analysis engine used for the different types of structures as well as the file path to these programs.

### Calculation Parameters

In this menu, the user may change the equivalent wind area calculations for non-dish antennas and setup the critical damping ratio "β" for vortex shedding analysis as required by CSA S37-01.



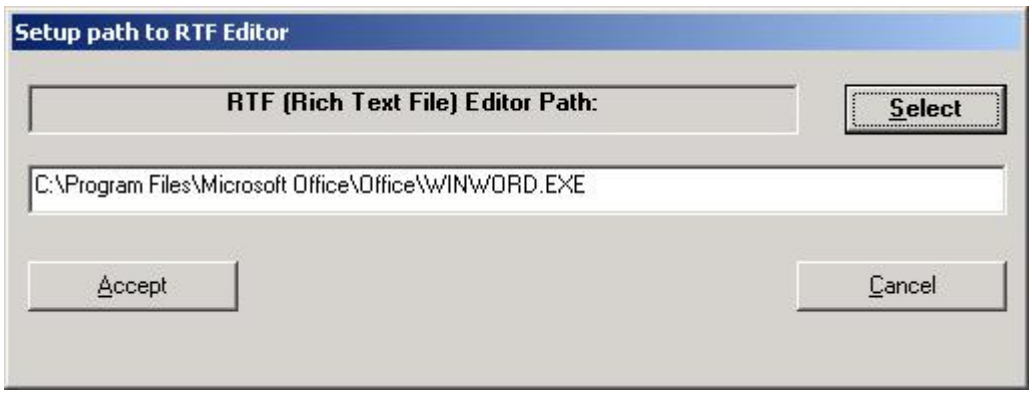

For non-dish antennas, the user can select the calculation method for the equivalent wind area of non-symmetric antennas.

Also, on this menu, the user (Engineer) can select the appropriate critical damping ratio β, the entered value impacts the calculations of the Vortex Shedding effects.

Note: it is possible that the program cannot perform the calculation of the dynamic effects of vortex shedding (an expression in formula, using square root of the difference between β and $C_2 * \rho * D^2 / M$ might be negative (Refer the National Building Code for explanation of the symbols used in the expression). If that happens then the program will display a warning message indicating that the reduced value of $C_g = 2.0$ can be used.

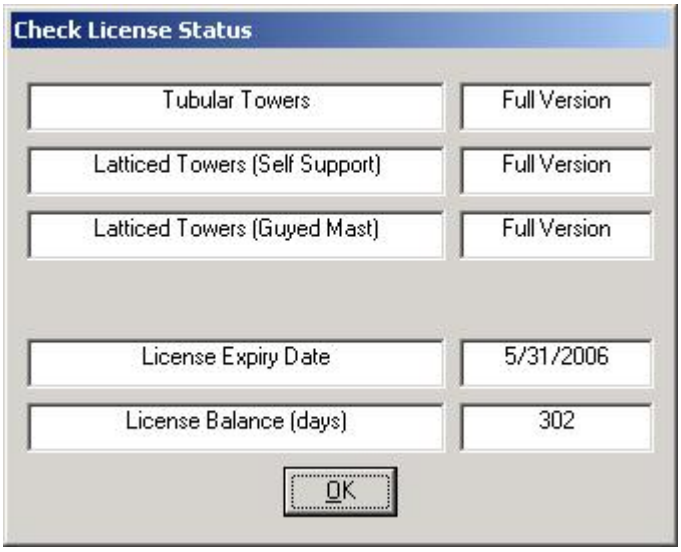
### Selection of Viewers

In these sub-menus, the user may select the programs and paths to the different viewer used by the program. For the .DXF files, .PDF and .RTF files. The user must select the path to the other 3rd-party programs that can used to open, view and print such files. An example of these sub-menus is shown below:




### Check License Status

In these sub-menus, the user may view the status of the license and the validity period as shown below:

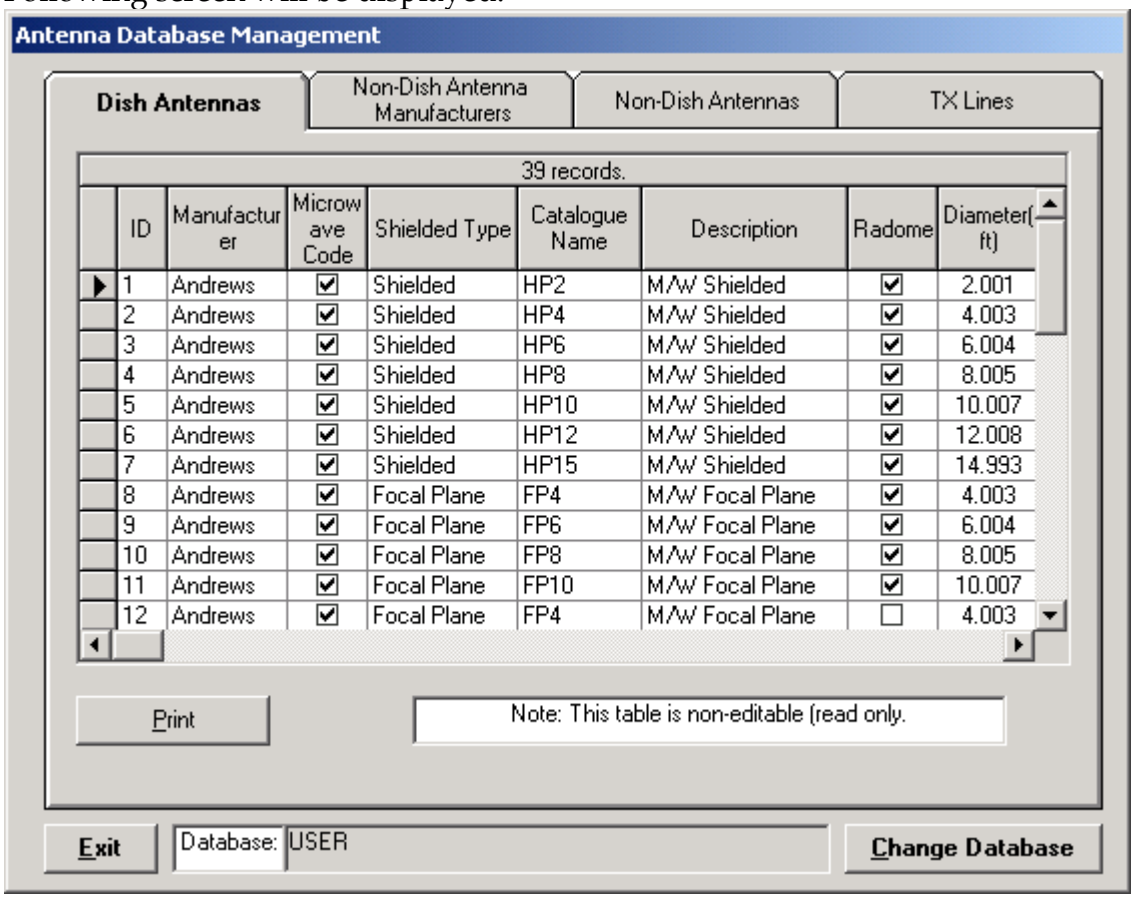


# Chapter 7 DATABASE MANAGEMENT

The program uses "USER" Antenna database for selection of the antennas, mounts and TX Lines.
Apart from the "USER" database there is "MASTER" database (not to be modified) and "REMOTE" database or databases, which can be placed in commonly accessed server and thus shared by different users typically within the same organization.
The "USER" and "REMOTE" databases can be modified by the users.

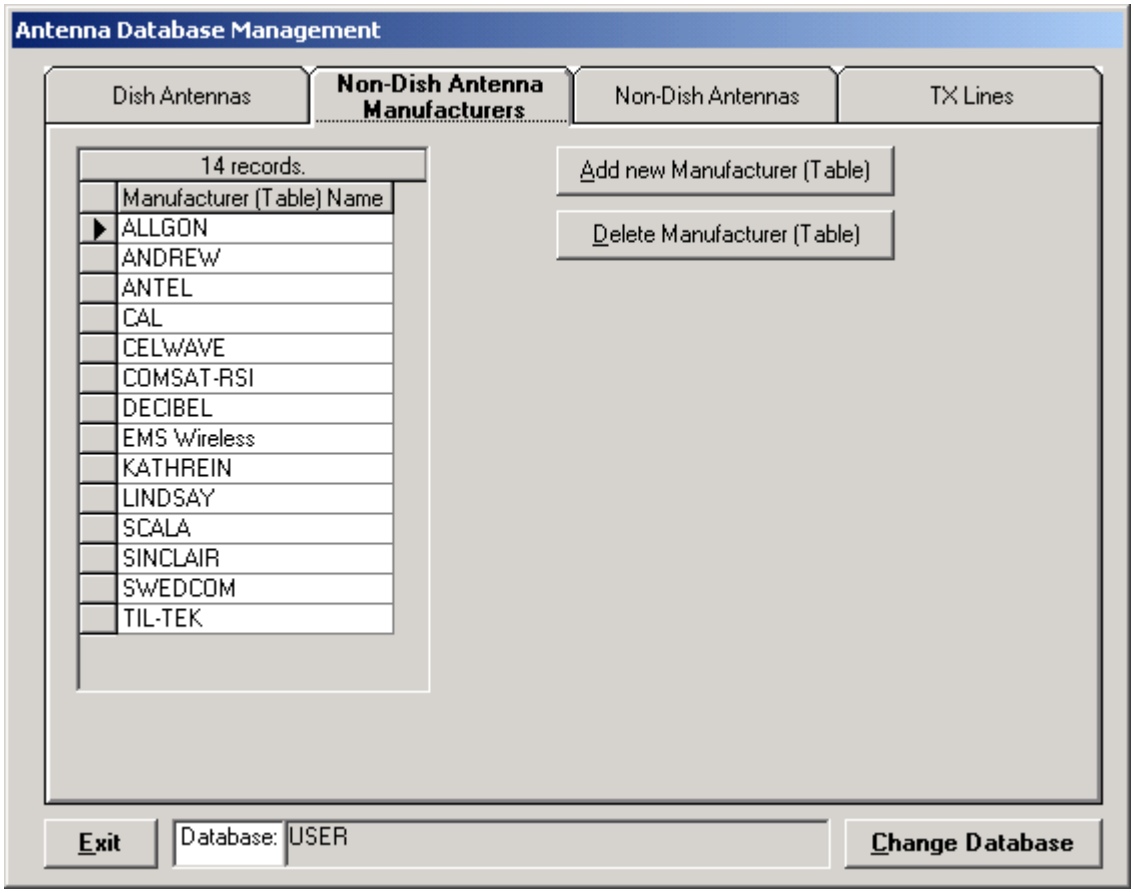
## Antennas Database

From the main menu, click on **Database Management** and **Antennas** submenu. Following screen will be displayed.



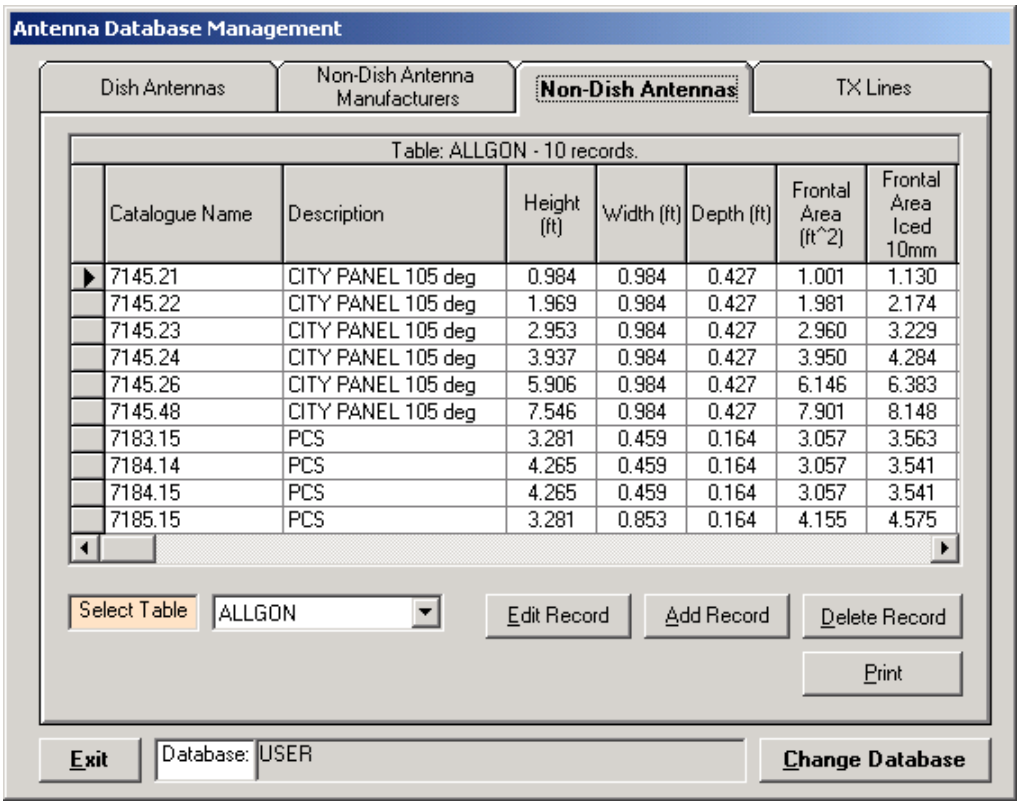

The first group "Dish Antennas" is non-editable (read only), as it covers practically all types and sizes of Microwave Dish Antennas.

### Non-Dish Antenna Manufacturers

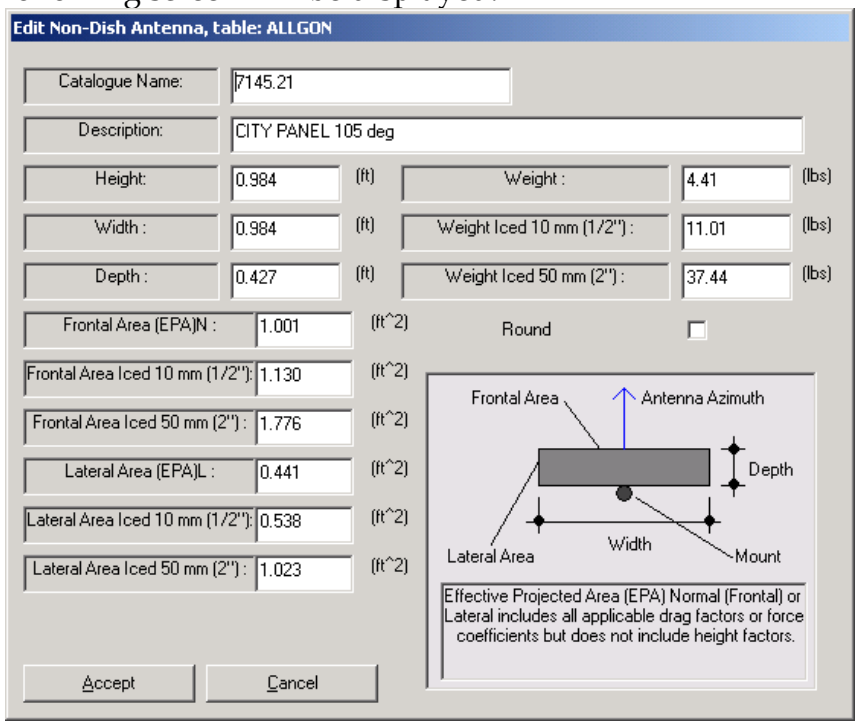



User can add new manufacturers. Such action will create an empty table and the user will then add records to such table. Deletion of manufacturer will cause removal of the manufacturer name from the list as well as deletion of a table of non-dish antennas associated with the manufacturer.
Default database is "USER" but any other "REMOTE" database can be selected (if present) and then the modifications are performed in the selected database. Once such "Remote" database is created and located remotely (on a server) other users can synchronize their local "USER" database with the "REMOTE".

## Non-Dish Antennas

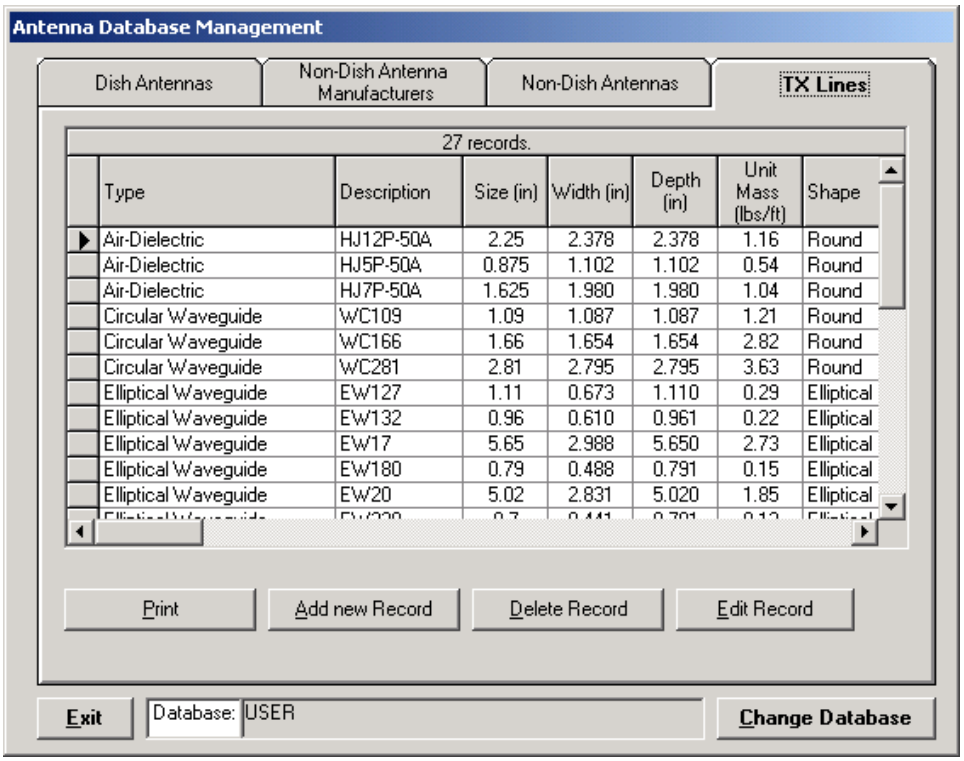



To edit or add record for selected table click on "Edit Record" or Add Record" buttons. Following screen will be displayed.

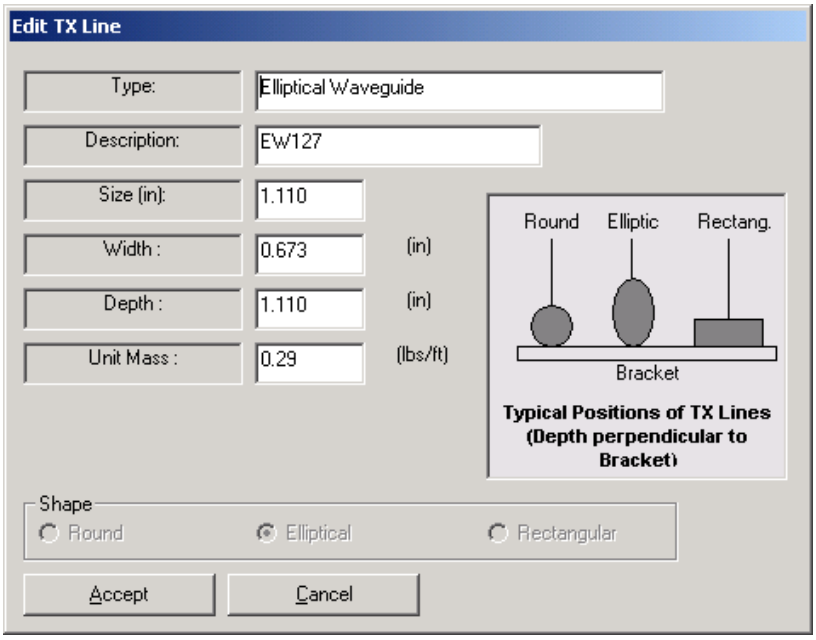



Changes of entries will become effective after "Accept" button is clicked.

## TX Lines



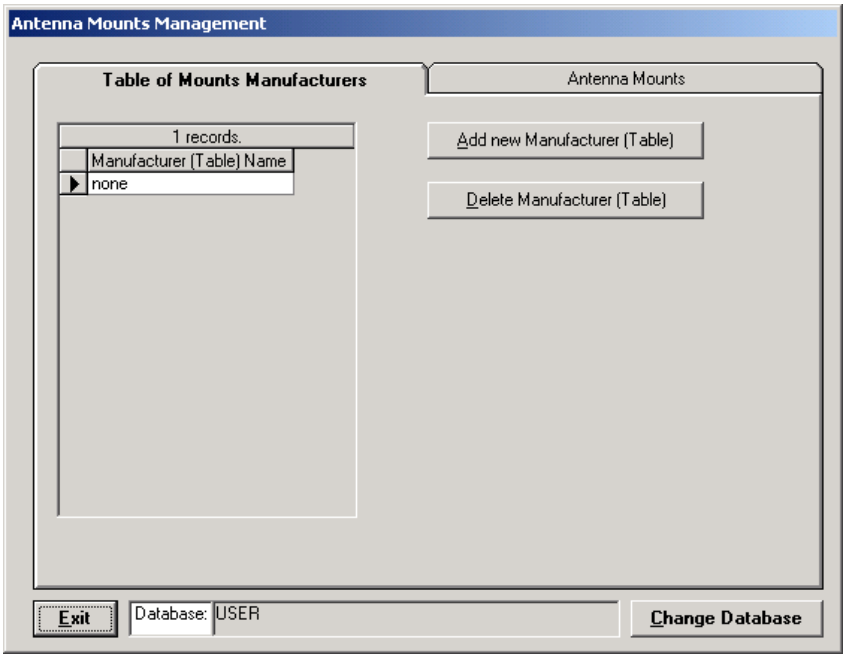

To add or edit a record click on "Add new Record" or "Edit Record" buttons respectively. Following screen will be displayed.




Changes of entries will become effective after "Accept" button is clicked.

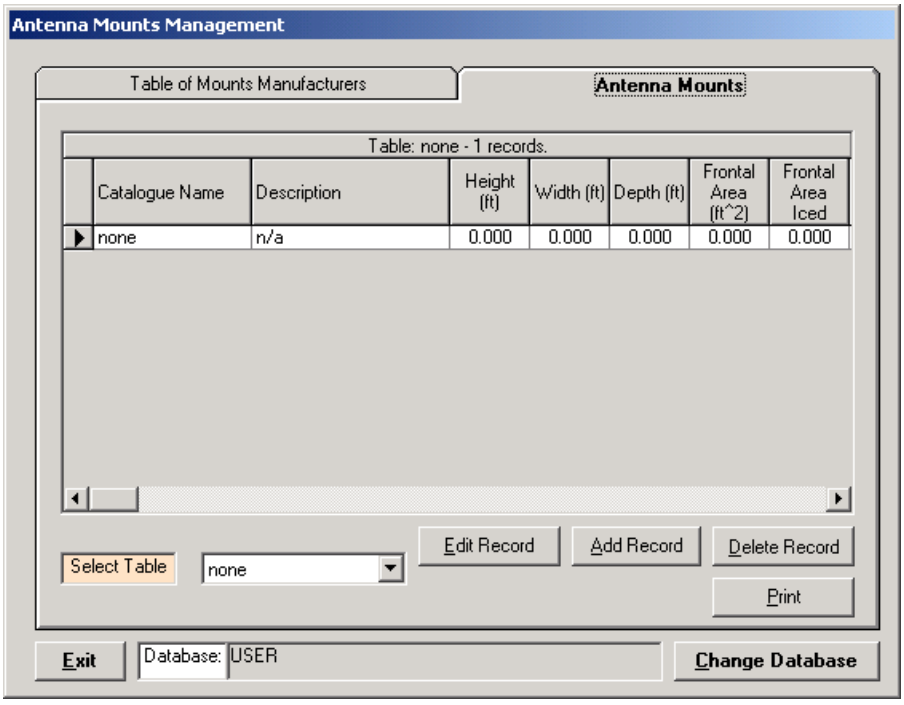
### Mounts

From the main menu, click on **Database Management** and **Mounts** submenu. Following screen will be displayed.

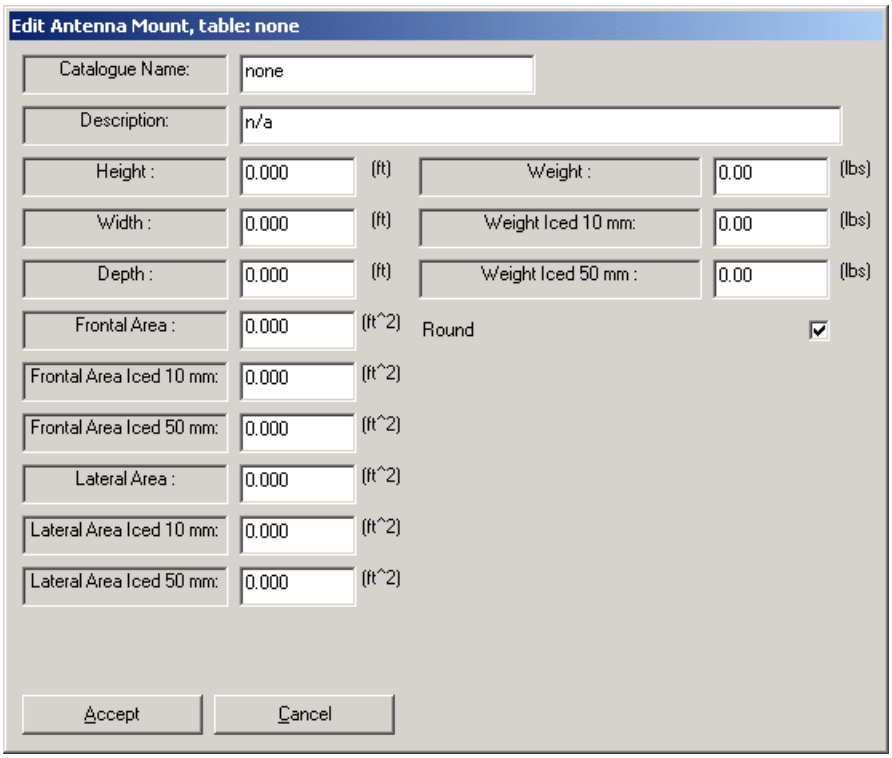



User may add new manufacturers (as is the case with Non-Dish Antennas) or delete them.

### Antenna Mounts



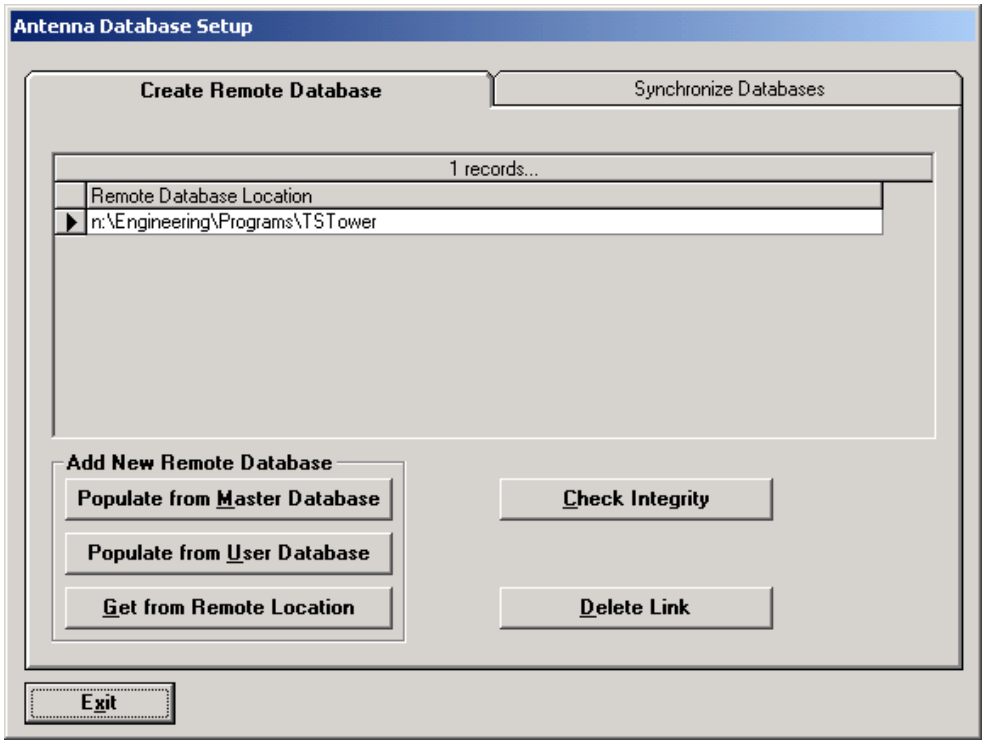
To add or edit a record click on "Add new Record" or "Edit Record" buttons respectively. Following screen will be displayed.




Perform the editing and press "Accept" button to update or "Cancel" otherwise.

### Database Setup

From the main menu, click on **Database Management** and **Database Setup** submenu.
Following screen will be displayed.

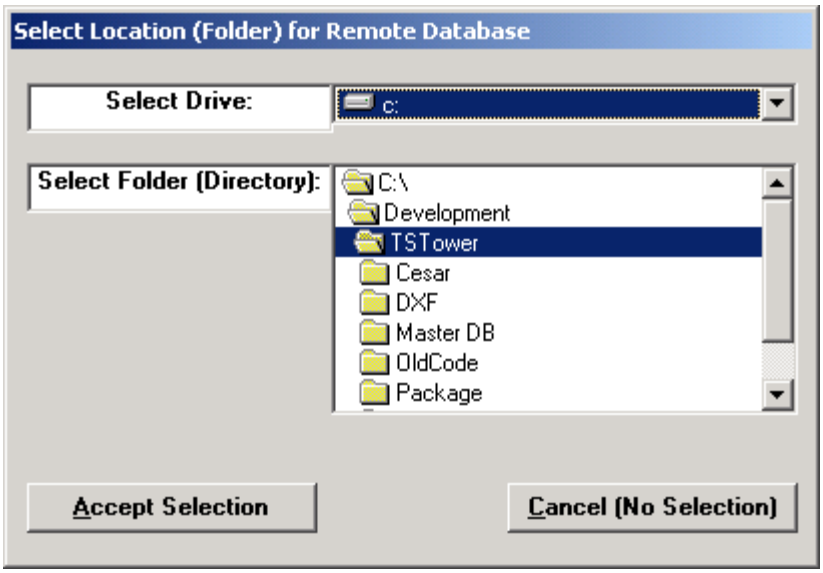



New remote database(s) can be created in three different ways:

"Populate from Master Database" – new "remote" database will be replicated from "Master Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

"Populate from User database" – new "remote" database will be replicated from "User Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

"Get from Remote Location" this action will prompt you to select existing "remote" database not linked to you computer so far.

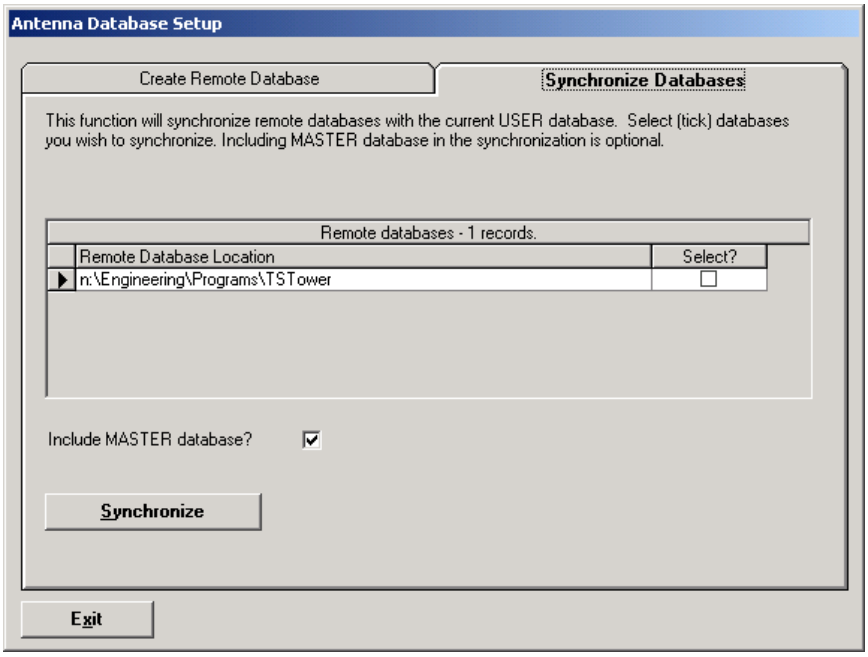



Once you selected the "remote" database it will be displayed in the "Remote Database Location" grid and you will have access to it via Database Management for Antennas or Mounts.

"Check Integrity" will perform the comparison of the records between selected remote database and "Master Database" and display differences.

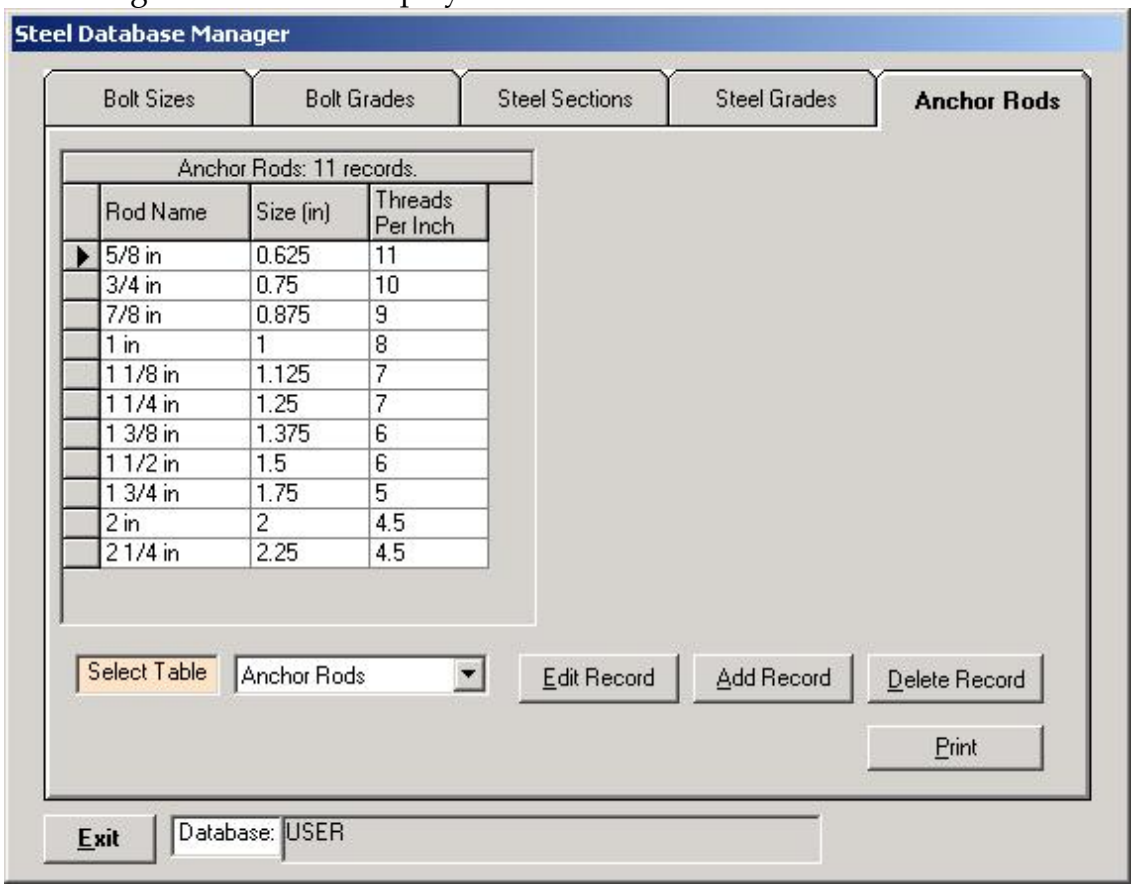
"Delete Link" will remove a link between selected "remote" database and the program (such database will not be accessible to you).

Synchronize Databases – instructions as shown on attached screenshot.



## Steel Database

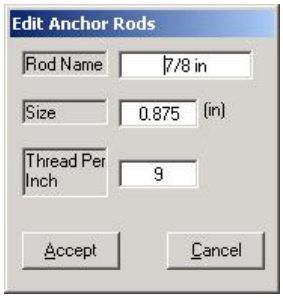
From the main menu, click on **Database Management** and **Steel DB** submenu. Following screen will be displayed.




For tubular structure (Poles), the user can edit anchor rod sizes as well as anchor rod grades.

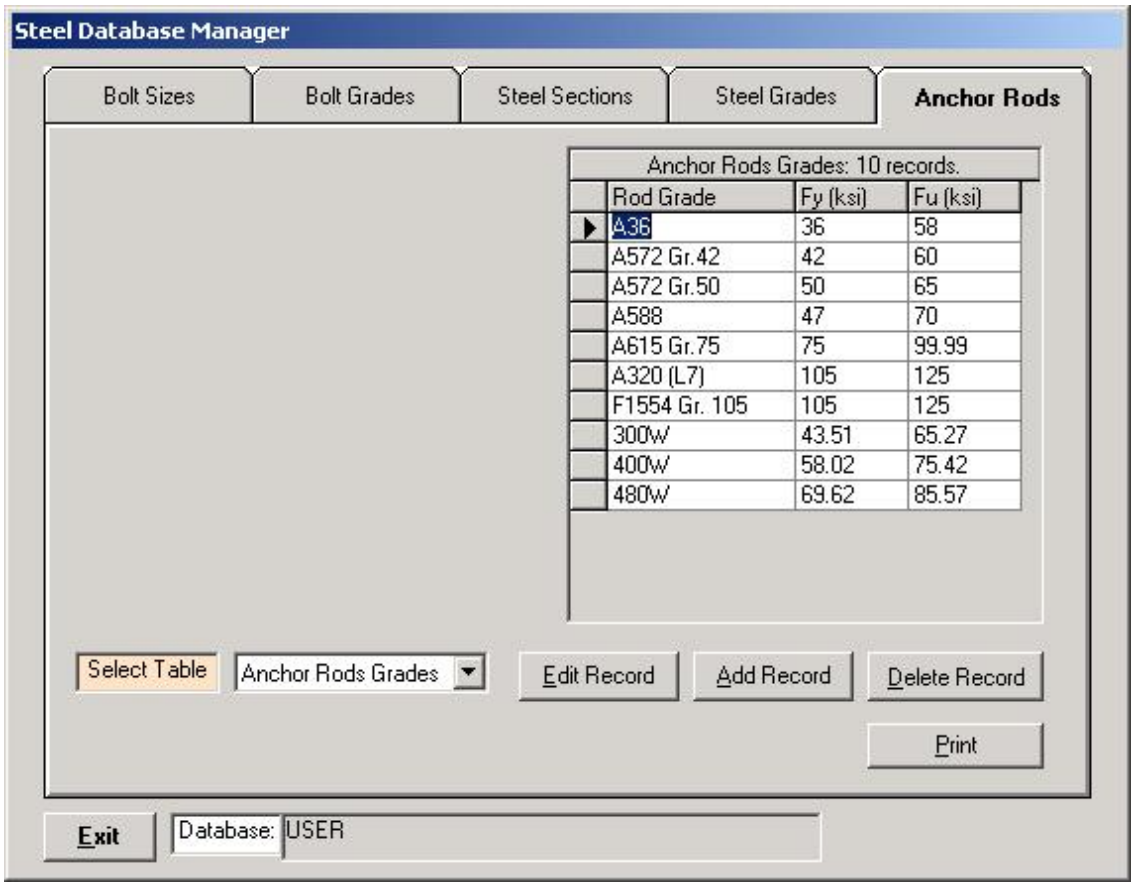
### Anchor Rods

In this table the user may view, add, edit or delete a record in the database for the anchor rod sizes. If a record is selected for editing, the following menu will show allowing the user to edit these values.

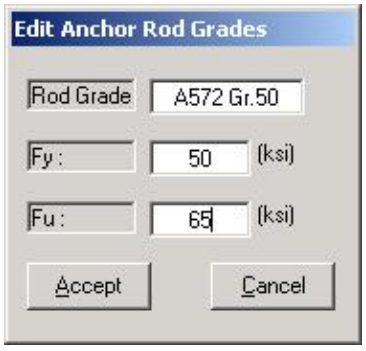


**Anchor Rods Grades**

By selecting to view the anchor grades from the Anchor rods tab, the following window will be displayed.

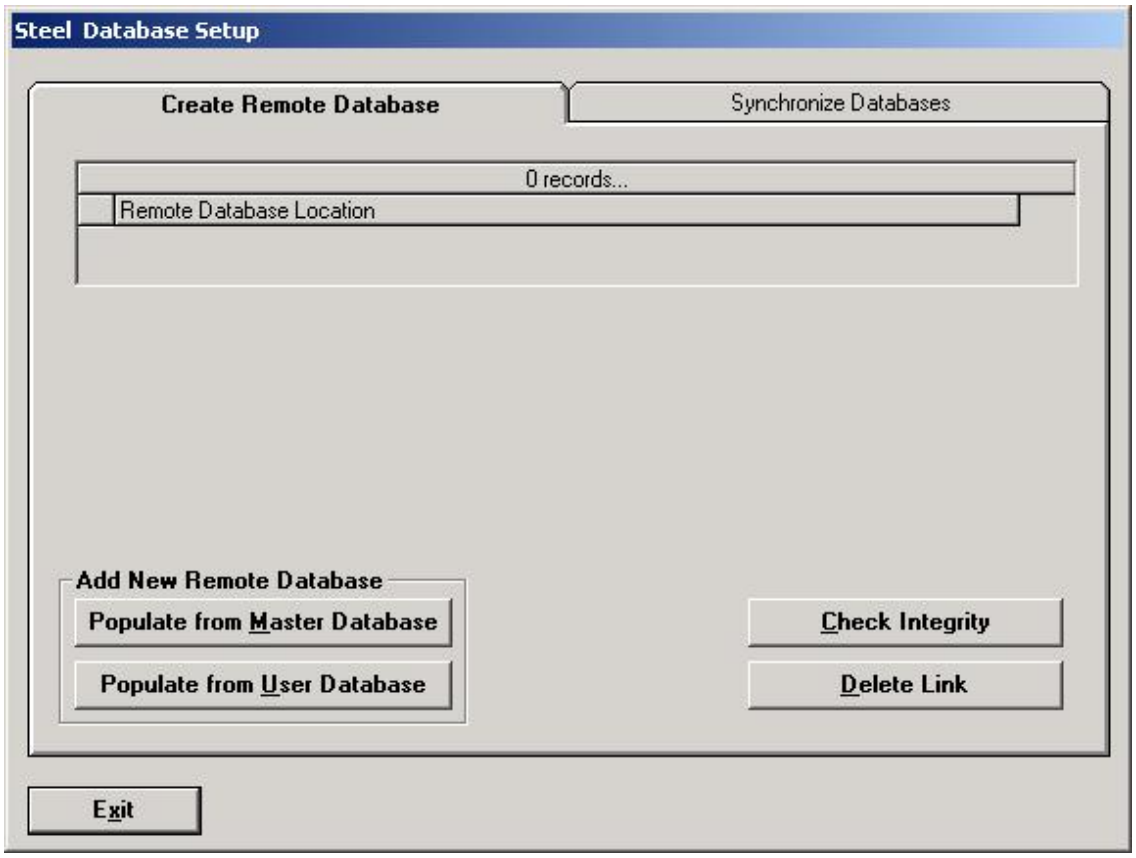



In this window the user may view, add, edit or delete a record in the database for the anchor rod sizes. If a record is selected for editing, the following menu will show allowing the user to edit these values.



### Database Setup

From the main menu, click on **Database Management** and then select **Steel DB** submenu and then **Database Setup** the following screen will be displayed.

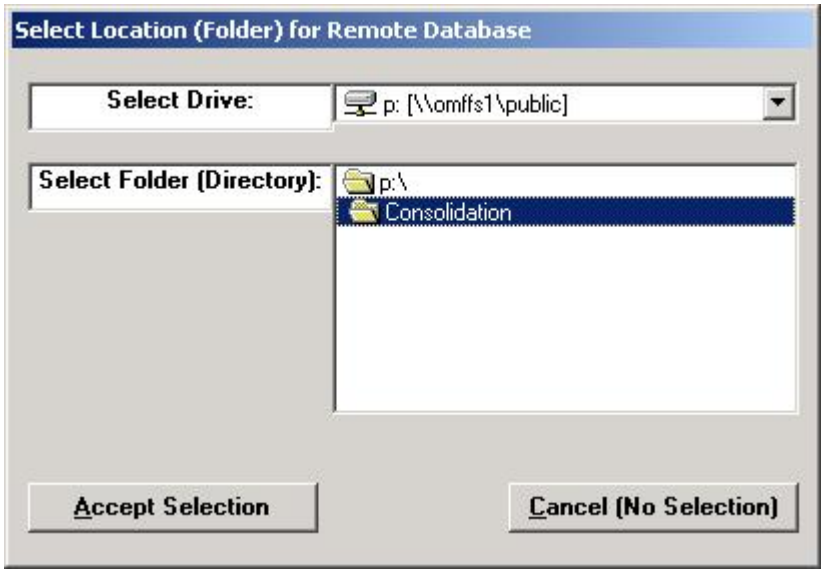



New remote database(s) can be created in three different ways:

"Populate from Master Database" – new "remote" database will be replicated from "Master Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

"Populate from User database" – new "remote" database will be replicated from "User Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

"Get from Remote Location" this action will prompt you to select existing "remote" database not linked to you computer so far.

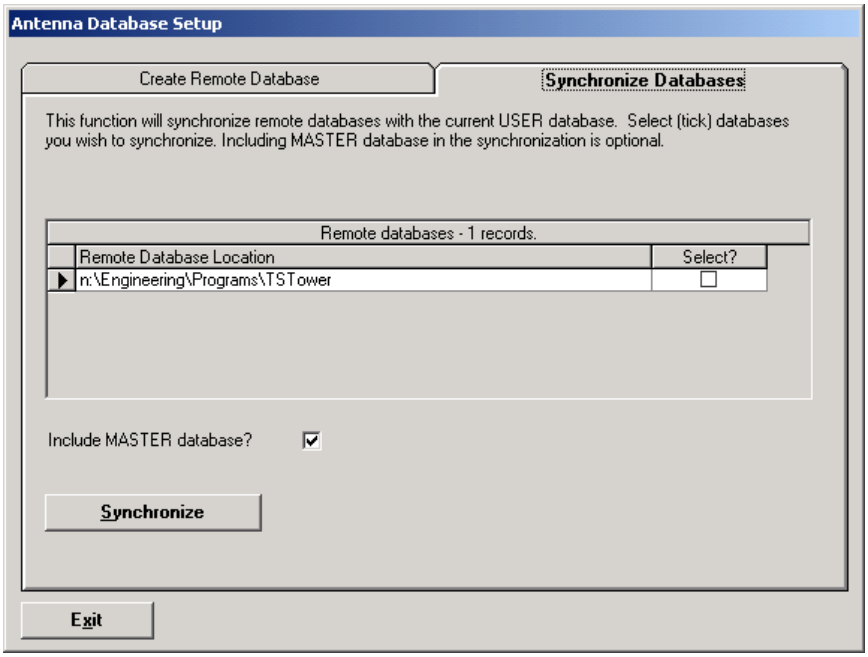



Once you selected the "remote" database it will be displayed in the "Remote Database Location" grid and you will have access to it via Database Management for Steel.

"Check Integrity" will perform the comparison of the records between selected remote database and "Master Database" and display differences.

"Delete Link" will remove the link between selected "remote" database and the program (such database will not be accessible to you).

Synchronize Databases – instructions as shown on attached screenshot.


Antenna Database Setup
Create Remote Database
Synchronize Databases
This function will synchronize remote databases with the current USER database. Select (tick) databases you wish to synchronize. Including MASTER database in the synchronization is optional.
Remote databases - 1 records.
Remote Database Location
Select?
n:\Engineering\Programs\TSTower
Include MASTER database?
Synchronize
Exit