

**Table of Contents**

<b><u>Section</u></b>	<b><u>Page</u></b>
25.1 GENERAL.....	25.1(1)
25.1.1 <u>Introduction</u> .....	25.1(1)
25.1.2 <u>Chapter Content</u> .....	25.1(1)
25.1.3 <u>Access to MDT Programs</u> .....	25.1(1)
25.2 MDT PROGRAMS.....	25.2(1)
Bridge End Stations.....	25.2(2)
Wingwall Lengths on Bents with Skew.....	25.2(6)
COGO87.....	25.2(7)
10 <sup>th</sup> Points Program.....	25.2(8)
D Depths for Spans on a Vertical Curve.....	25.2(9)
Vertical Curve Elevations on CL of Tangent Roadways with Grade Constant.....	25.2(11)
Bridge Beam Seat and Profile Grade Elevations.....	25.2(12)
Prestress Beam Design — Metric AASHTO 16 <sup>th</sup> .....	25.2(13)
Prestress Beam Design — Metric LRFD.....	25.2(14)
Prestressed Bulb-‘T’ Beam Design.....	25.2(15)
Prestressed Tri-deck Beam Design.....	25.2(17)
Slab Design.....	25.2(18)
Distribution of Flexural Steel.....	25.2(19)
Reactions at End and Intermediate Bents.....	25.2(21)
Round Column Design Using Strength Methods.....	25.2(22)
Rectangular Column, Biaxial Bending — Strength Design.....	25.2(24)
Soil Spring and Footing Stiffness.....	25.2(26)
Slab Bar Lengths on a Curved Bridge.....	25.2(27)
Skewed Slab Bar Lengths.....	25.2(29)
Reinforcing Steel Weight.....	25.2(31)
Prestressed Beam Diaphragm Volume.....	25.2(33)
End Bent Quantities.....	25.2(34)
Area Centroid Moment of Inertia.....	25.2(37)
25.3 EXTERNAL PROGRAMS.....	25.3(1)
AISIBEAM.....	25.3(2)
AISIsplICE.....	25.3(3)
BRASS-GIRDER(LRFD) <sup>TM</sup> .....	25.3(4)
BT Beam.....	25.3(5)
CONSPAN LA <sup>TM</sup> .....	25.3(6)
Florida Pier.....	25.3(7)
GEOPAK Bridge.....	25.3(8)
LPILE Plus.....	25.3(9)
Mathcad®.....	25.3(10)
MERLIN-DASH.....	25.3(11)
Opis.....	25.3(12)
Programmers File Editor (PFE).....	25.3(13)

**Table of Contents**

(Continued)

<b><u>Section</u></b>	<b><u>Page</u></b>
Pontis .....	25.3(14)
SEISAB.....	25.3(15)
SIMON Systems .....	25.3(16)
System VANCK .....	25.3(17)
Virtis .....	25.3(18)

## Chapter Twenty-Five

# COMPUTER PROGRAMS

### 25.1 GENERAL

#### 25.1.1 Introduction

The Bridge Bureau uses many computer programs for structural design, which can provide significant benefits. These include the capability of quickly analyzing several alternative designs (i.e., simulation capabilities), of reducing the probability of mathematical errors and for saving time by avoiding laborious hand calculations. However, the user of any computer program must consider the following:

1. Judgment and experience are critical to the proper interpretation of the computer outputs.
2. The user should, after the computer run, recheck inputs for accuracy.
3. The user should carefully check all output to ensure that answers are reasonable and logical and that there are no obvious errors. The check should include an equilibrium check in structural applications, for example, verifying that the sum of the applied loads equals the sum of the reactions.
4. The user should be familiar with the advantages and limitations of each program.

#### 25.1.2 Chapter Content

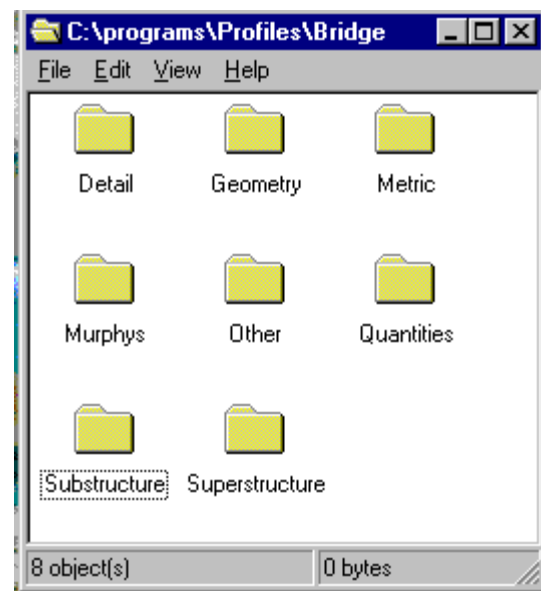
Chapter Twenty-five presents the majority of the computer programs used by the Bridge Bureau. For most programs, the Chapter provides a brief description of the program's applications, its inputs and its outputs. The Chapter is segregated into those programs that have been developed internally by the Department and those programs developed external to the Department.

#### 25.1.3 Access to MDT Programs

The MDT Library of Computer Programs is accessed through the following icon on most of the Bureau's Computers:



If you do not have this icon, the user can access the Library by clicking on "Start" then "Programs." Look for the Bridge Program listing in the column of options. By clicking on that icon, the user can call up the following selection of available folders:



Chapter Twenty-five describes most of the commonly used programs in these folders. For information on programs not described in Chapter Twenty-five, contact the Bridge Area Engineer.

## 25.2 MDT PROGRAMS

See Figure 25.2A.

Name	Location
1. Bridge End Stations	Metric Version - Bridge Programs\Metric\Bridge End Stations English Version - Bridge Programs\Geometry\ Bridge End Stations
2. Wingwall Lengths on Bents with Skew	Bridge Programs\Geometry\Wingwall Lengths on Bents with Skew ( <i>English Units</i> ) Bridge Programs\Metric\Wingwall Lengths on Bents with Skew
3. COGO87	Bridge Programs\Geometry
4. 10 <sup>th</sup> Points Program	Bridge Programs\Geometry
5. D Depths for Spans on a Vert. Curve	Metric – Bridge Programs\Metric\D Depths for spans on a Vert. Curve English – Bridge Programs\Geometry\D Depths for spans on a Vert. Curve
6. Vertical Curve Elevations on CL of Tangent Roadways with Grade Constant	Bridge Programs\Metric\Vertical Curve Elevations
7. Bridge Beam Seat and Profile Grade Elevations	Bridge Programs\Geometry
8. Prestress Beam Design — Metric AASHTO 16 <sup>th</sup>	Bridge Programs\Metric\Prestressed Beam Design
9. Prestress Beam Design — Metric LRFD	Bridge Programs\Metric\LRFD Prestressed Beam Design
10. Prestressed Bulb-‘T’ Beam Design	Bridge Programs\Superstructure\Prestressed Bulb-Tee Beam Design
11. Prestressed Tri-deck Beam Design	Bridge Programs\Superstructure\Tri Deck
12. Slab Design	Bridge Programs\Superstructure\Slab Design
13. Distribution of Flexural Steel	Metric – Bridge Programs\Metric\Distribution of Flexural Steel English – Bridge Programs\Substructure\Distribution of Flexural Steel
14. Reactions at End and Intermediate Bents	Bridge Programs\Metric\Reactions at End and Intermediate Bents
15. Round Column Design Using Strength Methods	
16. Rectangular Column, Biaxial Bending — Strength Design	Bridge Programs\Substructure
17. Soil Spring and Footing Stiffness	Bridge Programs\Other\footing.exe
18. Slab Bar Lengths on a Curved Bridge	Bridge Programs\Metric\Curved Slab Bar Lengths
19. Skewed Slab Bar Lengths	Bridge Programs\Metric\Bar Lengths (Skewed Slab)
20. Reinforcing Steel Weight	Bridge Programs\Metric\Reinforcing Steel Weight
21. Prestressed Beam Diaphragm Volume	Bridge Programs\Metric\Diaphq2m
22. End Bent Quantities	Bridge Programs\Quantities\End Bent Quantities for English or Bridge Programs\Quantities\Metric\End Bent Quantities for Metric
23. Area Centroid Moment of Inertia	Bridge Programs\Geometry

### MDT COMPUTER PROGRAMS

Figure 25.2A

**Title: Bridge End Stations****Units:** Available in both Metric and English Units**Location:** Metric Version - Bridge Programs\Metric\Bridge End Stations  
English Version - Bridge Programs\Geometry\Bridge End Stations**Description:**

Program calculates the required structure length based on the feature crossed, fill slopes, bridge geometry and roadway vertical curve information. The program rounds the bridge length to the nearest meter or foot and calculates the Station and Profile Grade Elevation at the bridge ends and center. This program is very helpful in preliminary design to help determine a bridge type and size. If used for preliminary design, the input values below can be approximated.

**Input Screen:**

```

***** Program for Bridge End Stations. *****

*** METRIC INPUT ***

Stations @ X-ING: Over = 2          Under =
Skew Angles:      X-ING =          Structure =

Dist CL Under to Toe:      Left =          Right =
Dist CL Over to Spill Pt: Left =          Right =
CL BRG to Back of Backwall =

Grade Constants @ Top:    Left =          Right =
Grade Constants @ Bot:   Left =          Right =
Fill Slope =             :1

UC Data for Over:  G1 =          G2 =
                   PI Sta =      PI Elev =
                   UC Length =

UC Data for Under: G1 =          G2 =
                   PI Sta =      PI Elev =
                   UC Length =

Input in STATIONS.

```

**Input Definitions:**

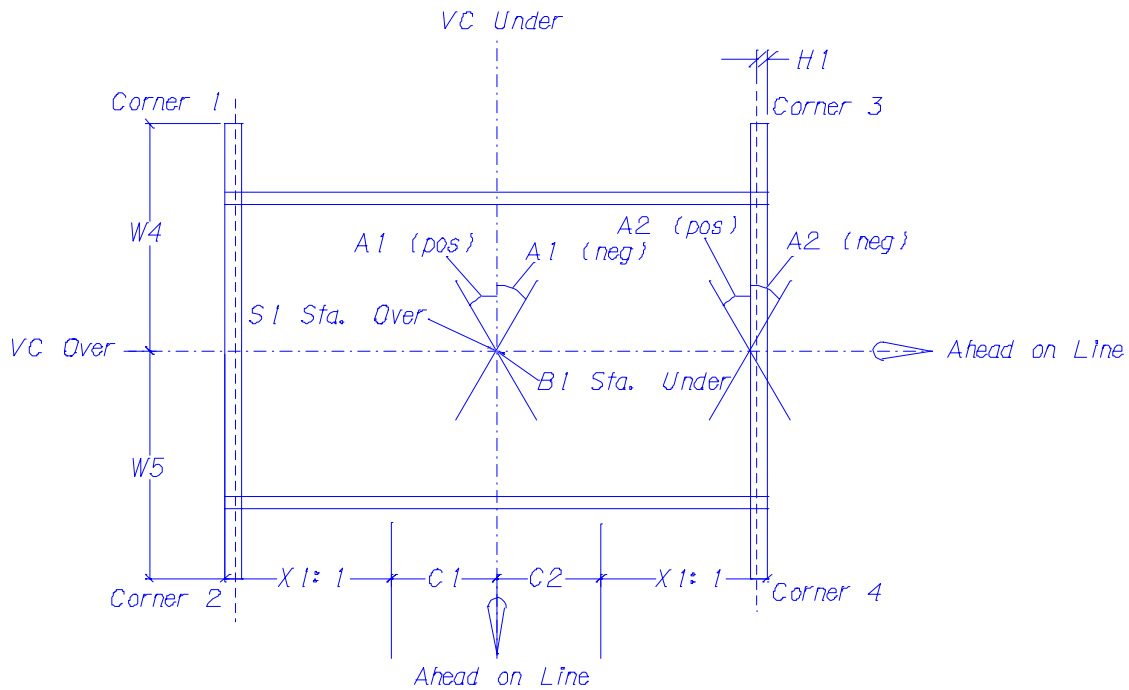
Terms:

- Over: Bridge that is being sized.
- Under: This could be a highway or interstate road, railroad tracks or stream crossing that the bridge (Over) is spanning.

Required information: A wing length based on the estimated beam type must be determined to calculate some of the input values below. The Wingwall Length program can be used.

S1 = Station @ Crossing Over = \_\_\_\_\_ Stations (ex: 2+50.50 = 2.5050 Stations)

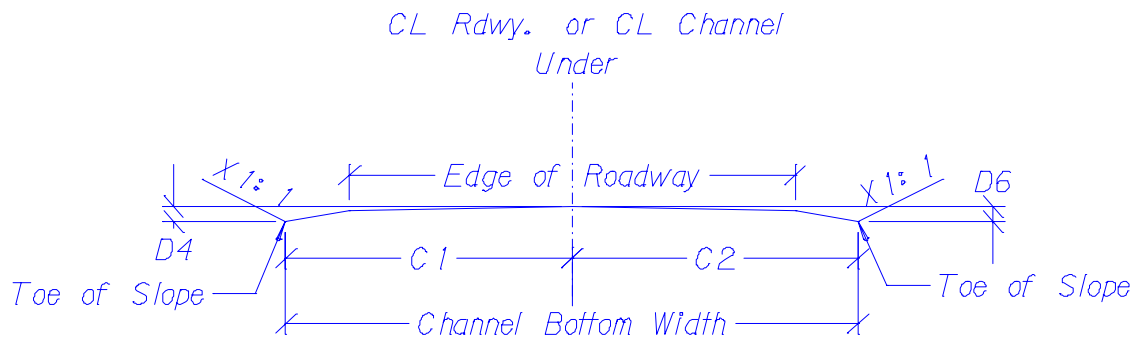
B1 = Station @ Crossing Under = \_\_\_\_\_ Stations



Skew angle positive if right side is ahead on line, negative if left side is ahead on line.

A1 = Skew angle of Crossing Under = \_\_\_\_\_ degrees

A2 = Skew angle of Structure Over = \_\_\_\_\_ degrees



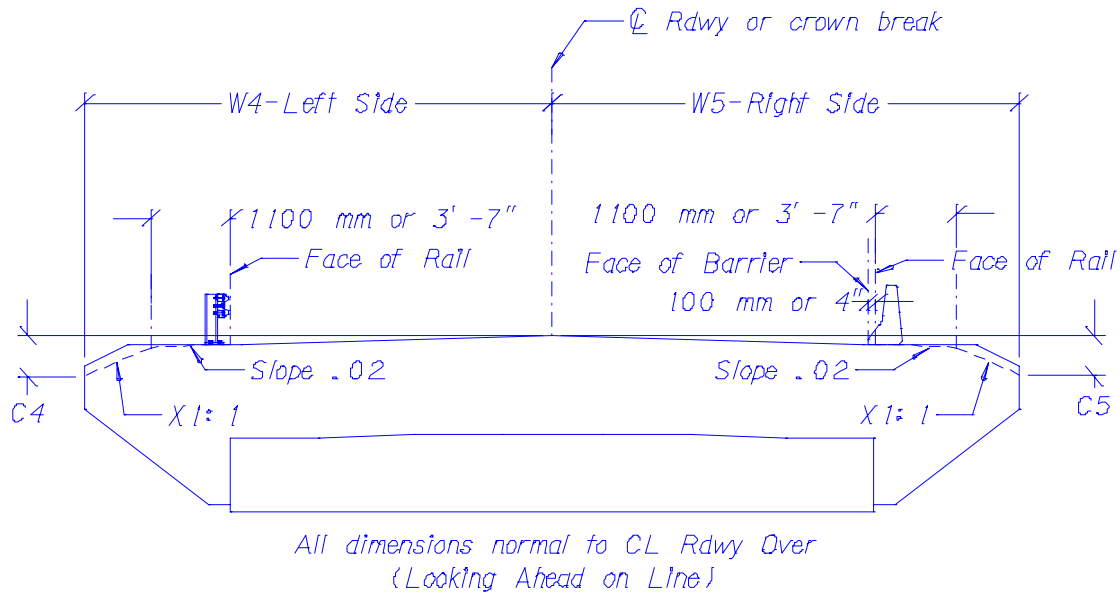
C1 and C2 are measured normal to  $\perp$  Under.

Roadway/Railroad: Measure from the Toe of Slope to  $\perp$  for each side

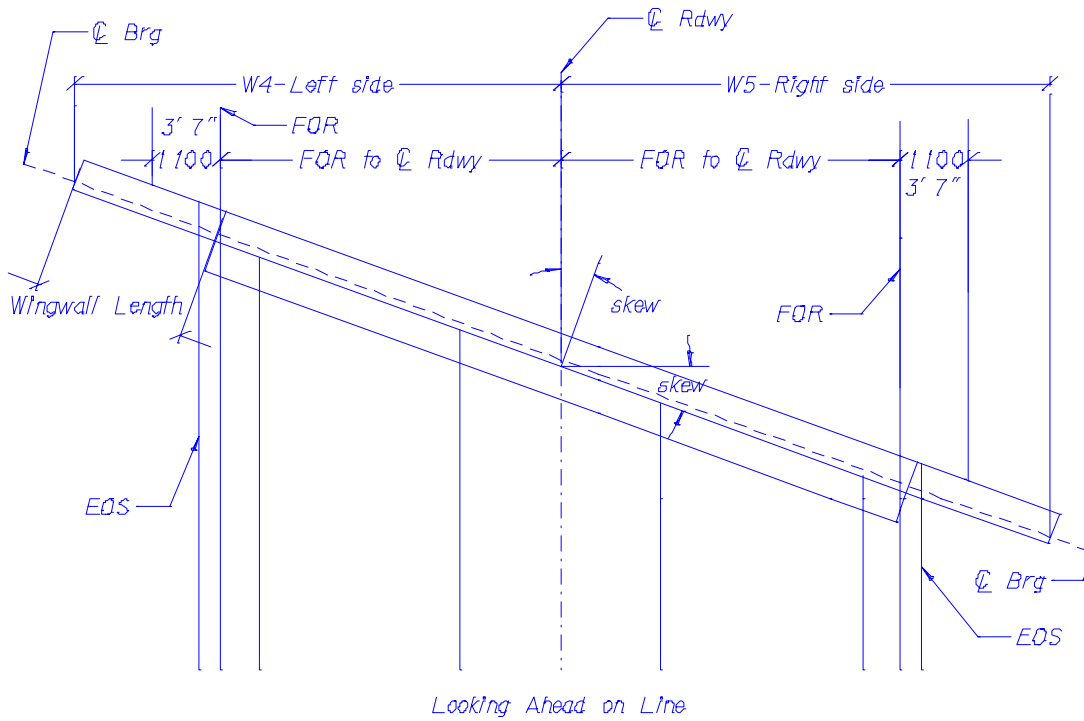
Stream crossing: C1 and C2 are typically half the channel bottom width (Hyd Report)

C1 = Left distance from  $\perp$  under to toe = \_\_\_\_\_ m or ft (Always positive)

C2 = Right distance from  $\perp$  under to toe = \_\_\_\_\_ m or ft (Always positive)



**For Skewed Bridges:**



W4 = Left perpendicular distance from  $\text{CL}$  over to spill point = \_\_\_\_\_ m or ft (Usually neg.)

W5 = Right perpendicular distance from  $\text{CL}$  over to spill point = \_\_\_\_\_ m or ft (Usually pos.)

H1 =  $\text{C} \text{ Brg. to fill face of backwall} = \underline{\hspace{2cm}}$  m or ft (Usually 0.4 m or 1.25 ft)  
(Flat Slab = 180 mm or 0.5833 ft)

(To use program with turned back wings, enter H1 as the distance from  $\text{C} \text{ Brg}$  to back of the turned back wingwall. The dimension W4 and W5 will also change measured from  $\text{C} \text{ Rdwy}$  to the outside edge of wingwall.)

C4 and C5 below are measured at the edge of wingwall on the fill face of Backwall

C4 = Left grade constant @ top =  $\underline{\hspace{2cm}}$  m or ft \*

C5 = Right grade constant @ top =  $\underline{\hspace{2cm}}$  m or ft \*

\* Sign convention: Negative if elevation of spill point is below Profile Grade @  $\text{C} \text{ Roadway}$   
Positive if elevation of spill point is above Profile Grade @  $\text{C} \text{ Roadway}$

\*\*D4 = Left grade constant @ bottom =  $\underline{\hspace{2cm}}$  m or ft

\*\*D6 = Left grade constant @ bottom =  $\underline{\hspace{2cm}}$  m or ft

\*\* Sign convention: Negative if elevation of toe is below Profile Grade @  $\text{C} \text{ Roadway under}$ .  
Positive if elevation of toe is above Profile Grade @  $\text{C} \text{ Roadway under}$ .

X1 = Fill Slope =  $\underline{\hspace{2cm}}$  (Usually 2, for 2:1 slope)

Vertical Curve — Over

G1 = Grade between PC & PI =  $\underline{\hspace{2cm}}$  %

G2 = Grade between PI & PT =  $\underline{\hspace{2cm}}$  %

PI = Station @ PI =  $\underline{\hspace{2cm}}$  Station

E1 = Elevation @ PI =  $\underline{\hspace{2cm}}$  m or ft

L1 = Length of Vertical Curve =  $\underline{\hspace{2cm}}$  Stations (Ex: 300 m = 3.00)

Vertical Curve — Under \*

G1 = Grade between PC & PI =  $\underline{\hspace{2cm}}$  %

G2 = Grade between PI & PT =  $\underline{\hspace{2cm}}$  %

PI = Station @ PI =  $\underline{\hspace{2cm}}$  Station

E1 = Elevation @ PI =  $\underline{\hspace{2cm}}$  m or ft

L1 = Length of Vertical Curve =  $\underline{\hspace{2cm}}$  Stations

\* For a stream crossing, the above information can be taken from the hydraulic report. G1 and G2 — the Channel Slope converted to %. P1 — usually same as B1 above. E1 — the Channel Bottom Elevation. L1 — anything wider than the bridge abutments.

### Verification of Results:

Verify that the correct input values were used and the length of the bridge seems reasonable.



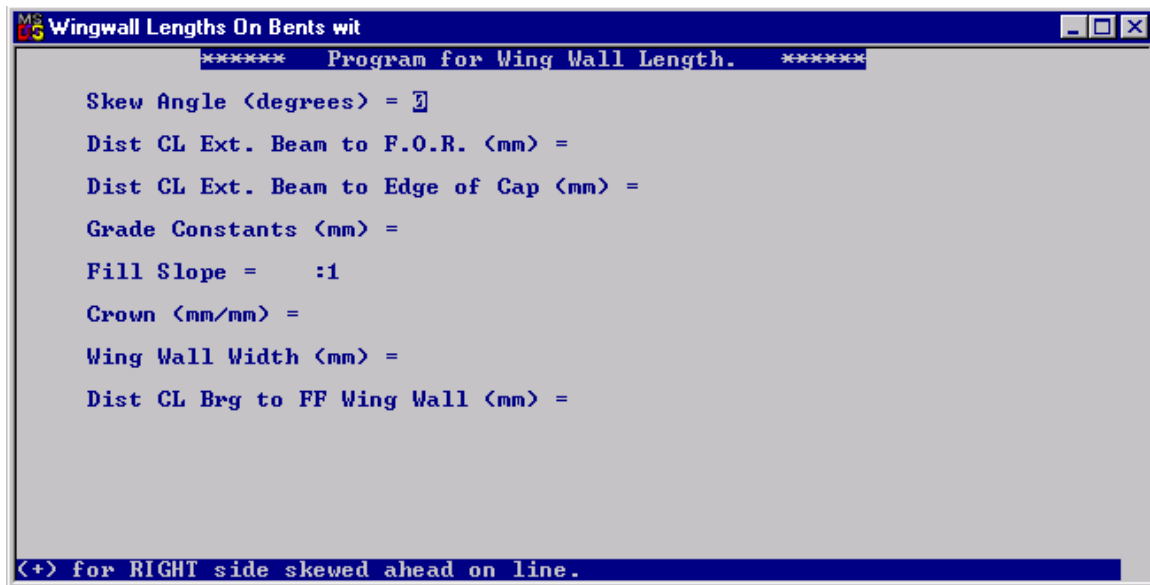
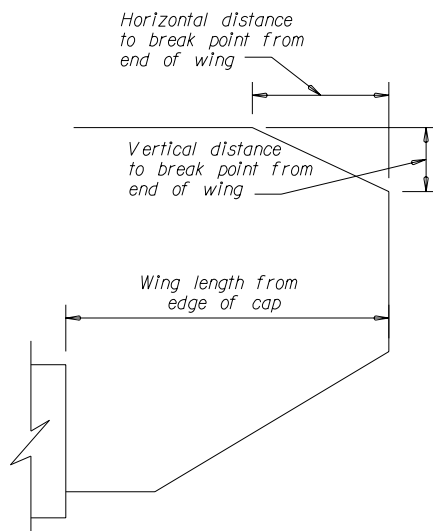
**Title: Wingwall Lengths on Bents with Skew**

**Location:** Bridge Programs\Geometry\Wingwall Lengths on Bents with Skew (*English units*)  
 Bridge Programs\Metric\Wingwall Lengths on Bents with Skew

**Description:** Calculates the wingwall length for a stub abutment.

**Input:**

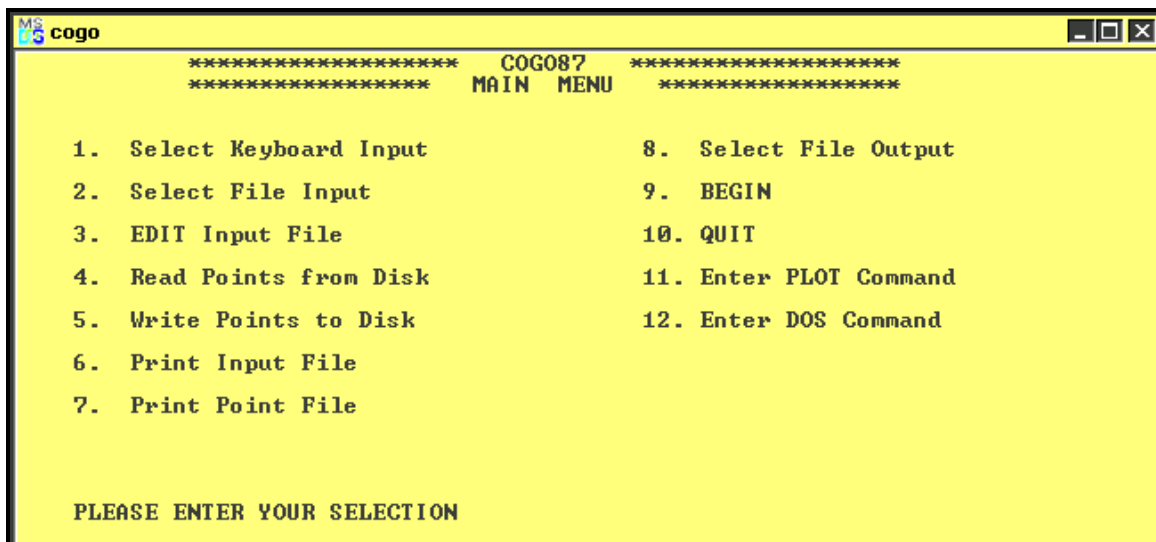
- Skew angle (left ahead positive, right ahead negative).
- Perpendicular distance from centerline of exterior beam to face of rail (FOR).
- Distance along the skew from centerline of exterior beam to the end of the cap.
- Grade constant (distance from top of slab at centerline exterior beam to top of cap).
- Fill slope (? :1).
- Crown or superelevation (m/m slope upward from exterior beam to centerline roadway is positive, downward slope is negative).
- Wingwall "width" (actually thickness).
- Distance from centerline of bearing to the fill face of the backwall.

**Output:**

**Title: COGO87****Location:** Bridge Programs\Geometry**Description:**

Program calculates coordinates and stations for geometric layouts. Refer to the published Manual for further information.

An alternative program is GEOPAK COORDINATE GEOMETRY.

**Input:**A screenshot of a DOS-style window titled 'cogo'. The window has a yellow background and a black border. At the top, it displays '\*\*\*\*\* COGO87 \*\*\*\*\*' and '\*\*\*\*\* MAIN MENU \*\*\*\*\*'. Below this, there is a list of 12 numbered options arranged in two columns. At the bottom, it says 'PLEASE ENTER YOUR SELECTION'.

```
MS-DOS cogo
***** COGO87 *****
***** MAIN MENU *****

1.  Select Keyboard Input          8.  Select File Output
2.  Select File Input             9.  BEGIN
3.  EDIT Input File              10. QUIT
4.  Read Points from Disk        11. Enter PLOT Command
5.  Write Points to Disk        12. Enter DOS Command
6.  Print Input File
7.  Print Point File

PLEASE ENTER YOUR SELECTION
```

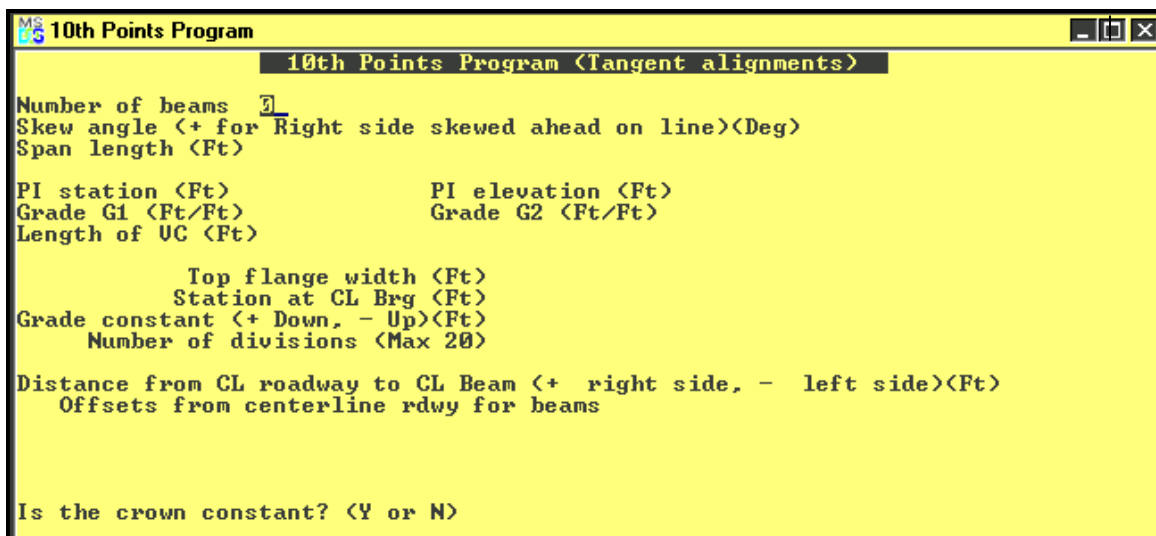
**Verification of Results:**

Verify that the correct input values were used. Check for reasonable results.

**Title: 10<sup>th</sup> Points Program****Location:** Bridge Programs\Geometry**Description:**

Program calculates elevations at 10<sup>th</sup> points for all girders within a span. Inputting the top flange width and haunch depth at the center of the beams allows elevations to be calculated at both sides of the top flange. This information is used during construction to set deck forms. Elevations can be calculated for a normal crown or superelevation transition. The program is limited to alignments on a horizontal tangent.

The program was written to work in feet but will work with meters because the units used in all calculations have consistent units. For the correct stations to be calculated, lengths should be adjusted for State Plane Coordinates.

**Input:**A screenshot of a DOS-style command window titled "10th Points Program". The window has a yellow background and a black border. The title bar includes the MS-DOS logo and standard window controls. The main area contains a list of input prompts for the program, such as "Number of beams", "Skew angle", "Span length", "PI station", "Grade G1", "Length of UC", "Top flange width", "Station at CL Brg", "Grade constant", "Number of divisions", "Distance from CL roadway to CL Beam", and "Is the crown constant?".

```
MS-DOS 10th Points Program
10th Points Program <Tangent alignments>
Number of beams  2
Skew angle <+ for Right side skewed ahead on line><Deg>
Span length <Ft>
PI station <Ft>          PI elevation <Ft>
Grade G1 <Ft/Ft>        Grade G2 <Ft/Ft>
Length of UC <Ft>
      Top flange width <Ft>
      Station at CL Brg <Ft>
Grade constant <+ Down, - Up><Ft>
      Number of divisions <Max 20>
Distance from CL roadway to CL Beam <+ right side, - left side><Ft>
      Offsets from centerline rdwy for beams
Is the crown constant? <Y or N>
```

**Verification of Results:**

Verify that the correct input values were used. Hand check an elevation output for verification of results.

**Title: D Depths for Spans on a Vert. Curve**

**Units:** Available in both Metric and English units.

**Location:** Metric – Bridge Programs\Metric\D Depths for spans on a Vert. Curve  
English – Bridge Programs\Geometry\D Depths for spans on a Vert. Curve

**Description:**

The output from this program is used to calculate the correction to the camber based on the vertical curve for concrete or steel girders. Output is determined as follows:

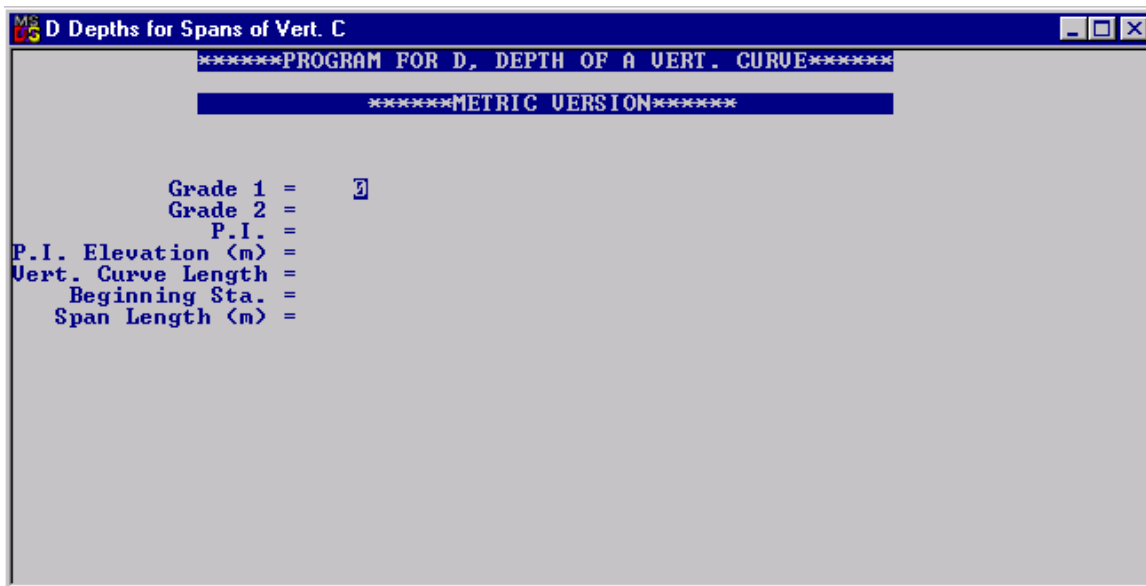
The program calculates elevations at 10<sup>th</sup> points along the string line within a specified span length. The string line is the straight line between the beginning and ending station entered into the program. At each 10<sup>th</sup> point, the difference between the Profile Grade Elevation and String Line Elevation is calculated.

The Difference (mm) is used to determine varying haunch depth for concrete and steel girders as well as field splice locations for steel girders.

When using this program for steel girder design, the user should provide a Table of Camber Information (see Structures Manual p. 5.4(26)). This table indicates the amount of camber, to the nearest millimeter, that will be necessary for each girder at the 10<sup>th</sup> points of each span and at the field splices.

A Girder Camber Diagram is also necessary to illustrate the location of the deflections for each tenth point on the span, the location of field splices, the string line slope and the vertical curve offset.

Finally, a Longitudinal Section diagram should be included in design. The Longitudinal Section illustrates an elevation view of the girder and bearing placement at each bent. It is important that the diagram include the offset dimension and the string line slope.

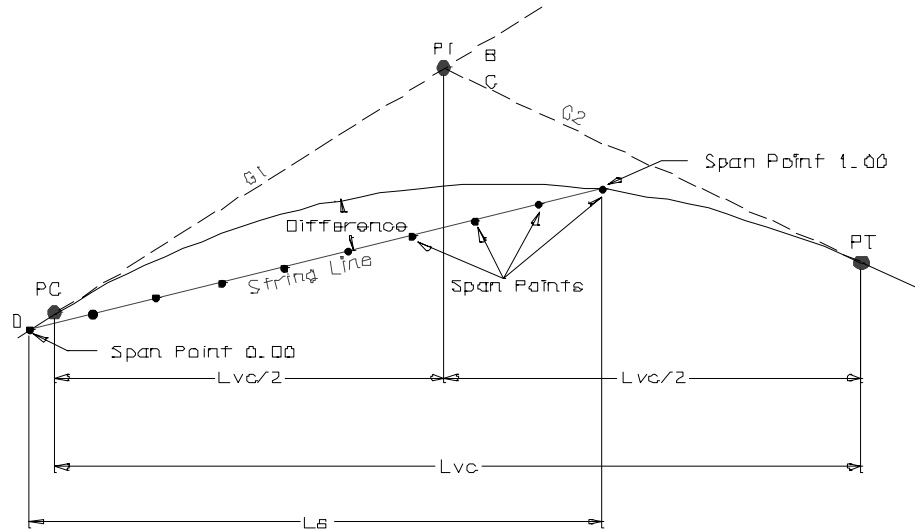
**Input:**

```
MS-DOS D Depths for Spans of Vert. C
*****PROGRAM FOR D, DEPTH OF A VERT. CURVE*****
*****METRIC VERSION*****

Grade 1 = 
Grade 2 = 
P.I. = 
P.I. Elevation (m) = 
Vert. Curve Length = 
Beginning Sta. = 
Span Length (m) =
```

**Output:**

Graphical Description of Input and Output:



- PC = Point of Curvature
- PI = Point of Intersection
- PT = Point of Tangency
  
- G1 = Grade between PC & PI = (%)
- G2 = Grade between PI & PT = (%)
- B = Station @ PI = (sta.)
- C = Elevation @ PI = (m)
- Lvc = Length of Vertical Curve = (sta.)
- D = Beginning Station = (sta.)
- Ls = Span Length = (m)

To calculate the String Line Slope, use the following equation:

$$\text{String Line Slope} = \left( \frac{[\text{Profile Grade Elev. @ (Span Pt. 1.0)}] - [\text{Profile Grade Elev. @ (Span Pt. 0.0)}]}{Ls} \right) * 100$$

**Verification of Results:**

Verify that the correct input values were used. Hand checks can be done for verification of results.

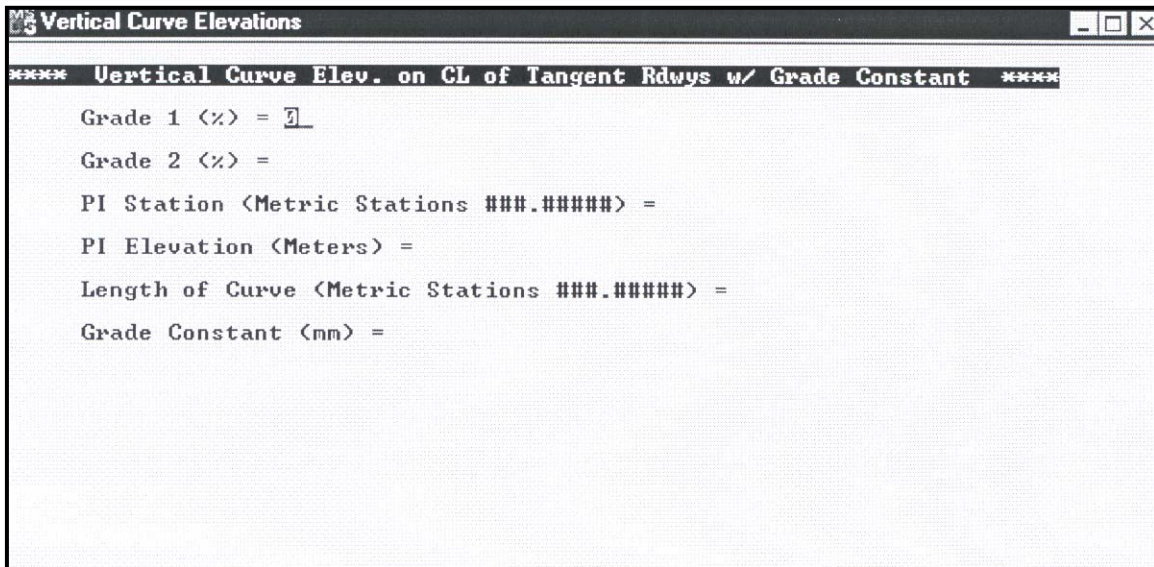
**Title: Vertical Curve Elevations on CL of Tangent Roadways with Grade Constant****Location:** Bridge Programs\Metric\Vertical Curve Elevations**Description:**

The program calculates vertical curve elevations at specific stations on a tangent alignment.

- Input the grade entering the curve (Grade 1) and the grade exiting (Grade 2) in percent (“+” for rising grade left to right and “-” for declining grade left to right).
- Input the vertical curve’s PI Station and Length in stations.
- Input the Grade Constant if needed. The grade constant will provide elevations at a constant distance above or below the gradeline input above.

Once the data is input, elevations can be displayed for specific stations for a series of elevations at constant increments and for a series of elevations at varying increments.

Choosing the “*Quit Application*” option from the menu displays selected elevations in a printable format.

**Input:**

```
**** Vertical Curve Elev. on CL of Tangent Rdwys w/ Grade Constant ****
Grade 1 (<math>\%</math>) = 0
Grade 2 (<math>\%</math>) =
PI Station (Metric Stations ###.#####) =
PI Elevation (Meters) =
Length of Curve (Metric Stations ###.#####) =
Grade Constant (mm) =
```

**Verification of Results:**

Verify that the correct input values were used. Code the program to output a point of known elevation or hand check a minimum of one elevation as output for verification of results.

**Title: Bridge Beam Seat & Profile Grade Elevations****Location:** Bridge Programs\Geometry**Description:**

Program calculates beam seat elevations for all girders within a span. Multiple spans can be input. Elevations can be calculated for a normal crown or superelevation transition. The program is limited to alignments on tangent in the horizontal plane.

The program was written to work in feet but will work with meters because the units used in all calculations have consistent units. For the correct stations to be calculated, lengths should be adjusted for State Plane Coordinates.

**Input:**

```

MS-DOS Bridge Beam Seat & Profile Gr
Beam Seat Elevations

Grade1 = 2          Grade2 =
PI Station =       PI Elevation =
UC Length =

Constant crown? <Y or N>      Crown =
Skew Angle =
Grade Constant =
Station @ CL Bearing =
Number of Beams =
Beam Locations <Lt(-) or Rt(+)> of CL Roadway <FT.>
1st          2nd          3rd          4th
5th          6th          7th          8th
9th          10th         11th         12th
13th         14th         15th         16th

Distance to next CL Bearing Station <FEET> =

Input Grade in PERCENT.
  
```

**Note:** Input guidance is provided on the bottom line of the menu.

**Verification of Results:**

Verify that the correct input values were used. Hand check an elevation output for verification of results.

**Title: Prestress Beam Design — Metric AASHTO 16th****Location:** Bridge Programs\Metric\Prestressed Beam Design**Description:**

The program designs Prestressed I-Girder and Non-standard sections in accordance with the AASHTO 1996 Standard Specifications, 16<sup>th</sup> Edition with the 1999 Interims. The program allows the designer to quickly determine the optimum girder spacing for a given span length, and it provides the typical design parameters needed for the erection plan. The program assumes and provides a strand pattern based on a 50-mm grid up to a maximum number for each girder consistent with the capacity of local suppliers. The program also is set to use concrete strengths consistent with the typical MDT allowable. The program will toggle between composite dead loads for both the T101 bridge rail and concrete barrier rail, and it includes the weight of a future wearing surface.

**Input:**

```

Metric Prestressed Beam Design Metric
Roadway Width (mm): 7
Span Length (mm):
Beam Spacing (mm):
Slab Thickness (mm):
Haunch Depth (mm):
Wearing Surface Thick (mm):
Composite Dead Load (kN/m):
Beam Strength (MPa):
Slab Strength (MPa):
Number of Inter Diaphs:
Live Load (1=HS20 & Mil, 0=HS20):
LowLax or Stress Relieved Strand (L or S):
Beam Type (Interior, Exterior, Both)(I, E or B):

Type 1
Type A
Type 4
M-72
Type 10A
MT-28
Non-Std

Usually 200 Less for Barrier
Widths: 7200 (24), 8400 (28), 9000 (30), 9600 (32), 10800 (36), 12000 (40)

```

*Note: Input Road Width, Span Length and assumed Beam spacing in mm. Input slab strength in MPa using regional parameters (28 or 31). The remaining input values will automatically default unless overridden.*

**Verification of Results:**

- Verify that the correct input values were used.
- Verify that the transfer strength will allow daily bed rollover.
- Verify that a possible strand pattern was found and is reasonable.
- Note the controlling design (Interior or Exterior) and denote the controlling design moment and stresses for the plans.
- If the trial design fails, revise the design and rerun another trial.



**Title: Prestress Beam Design — Metric LRFD****Location:** Bridge Programs\Metric\LRFD Prestressed Beam Design**Description:**

The program designs Prestressed I-Girder, Bulb-Tee, Tri-Deck and Non-standard sections in accordance with the AASHTO 1998 LRFD Specifications. The program allows the designer to quickly determine the optimum girder spacing for a given span length, and it provides the typical design parameters needed for the erection plan. The program assumes and provides a strand pattern based on a 50-mm grid up to a maximum number for each girder consistent with the capacity of local suppliers. The program also is set to use concrete strengths consistent with the typical MDT allowable. The program will toggle between composite dead loads for both the T101 bridge rail and concrete barrier rail, and it includes the weight of a future wearing surface.

**Input:**

```

Metric LRFD Prestressed Beam Design LRFD Metric
Roadway Width (mm): 0
Span Length (mm): 0
Beam Spacing (mm): 0

Slab Thickness (mm): 0
Haunch Depth (mm): 0
Wearing Surface Thick (mm): 35

Slab Overhang. (Assumed T-101)
Beam Embedment - Outside Edge (mm): 15
Thickness @ Edge of Slab (mm): 0
Edge of Slab to Face of Rail (mm): 350
Face of Barrier or Rail to CL Beam (mm): 0

Rail Wt (kN/m): 0
Future Overlay Wt (kN/m): 0
Beam Strength (MPa): 41
Slab Strength (MPa): 31
Number of Inter Diaphragms: 0
LowLax or Stress Relieved Strand (L or S): L
Beam Type (Interior, Exterior, Both)(I, E or B): B

Type 1
Type A
Type 4
M-72
Type 10A
MT-28
Bulb-Tee
Rib/Tri-Deck
Non-Std

Skew and ADIT
Skew Angle (Deg): 0
ADIT Per Direction: 5000

Widths: 7200 (24), 8400 (28), 9000 (30), 9600 (32), 10800 (36), 12000 (40)
Usually 200 Less for Barrier

```

*Note: Input Road Width, Span Length and assumed Beam spacing in mm. Input slab strength in MPa using regional parameters (28 or 31). The remaining input values will automatically default unless overridden.*

**Verification of Results:**

- Verify that the correct input values were used.
- Verify that the transfer strength will allow daily bed rollover.
- Verify that a possible strand pattern was found and is reasonable.
- Note the controlling design (Interior or Exterior) and denote the controlling design moment and stresses for the plans.
- If the trial design fails, revise the design and rerun another trial.

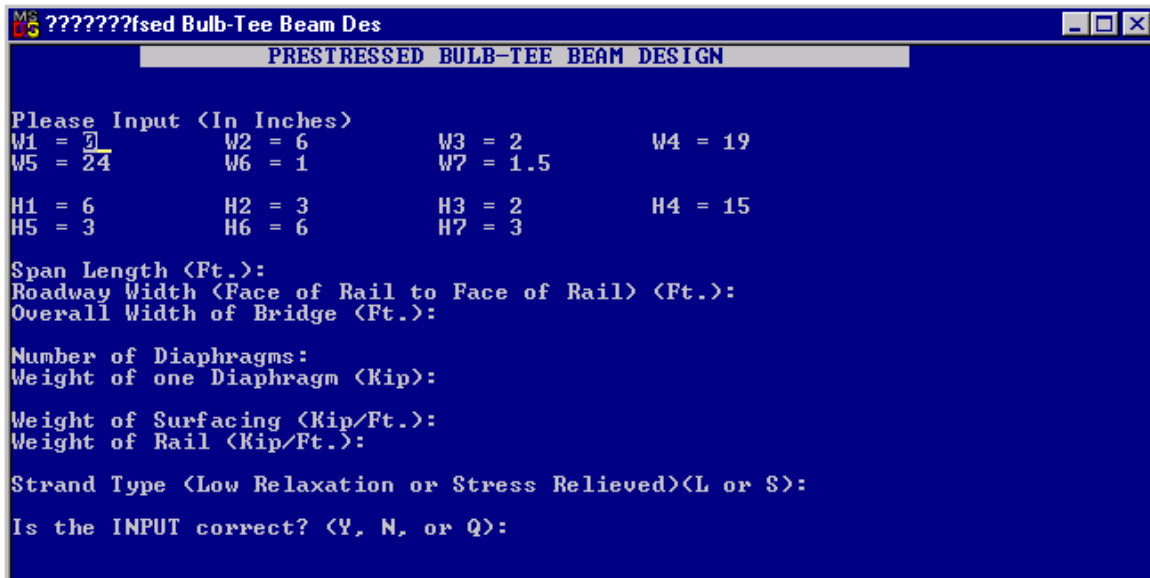
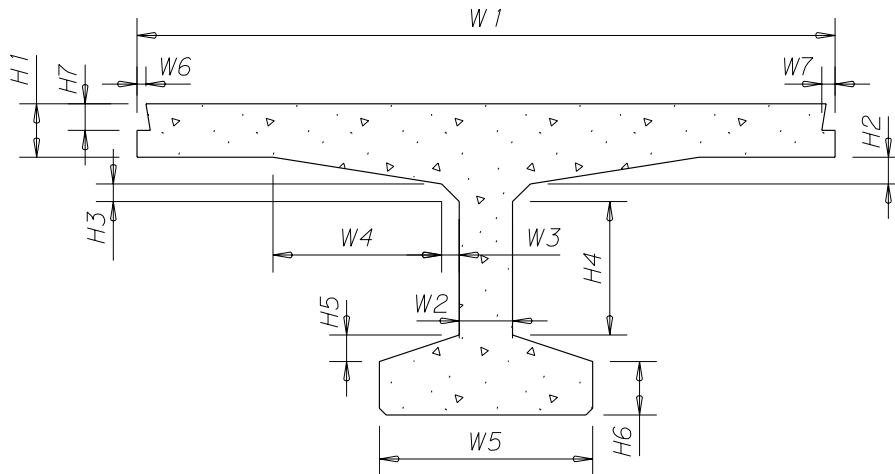
**Title: Prestressed Bulb-'T' Beam Design**

**Location:** Bridge Programs\Superstructure\Prestressed Bulb-Tee Beam Design

**Description:**

Given section dimensions and span requirements (see screen view below), the program designs the beam section necessary to meet them.

**Input:** Define the beam section by referring to the diagram.



**Output:**

The program provides section properties for the section (moment of inertia, section modulus, centroid location), dead and live load moments, dead load deflections, and allowable and actual stresses at transfer and in final configuration. The program generates stresses and deflections each for an interior beam and for an exterior one. It also generates a possible strand pattern for each one.

**Verification:**

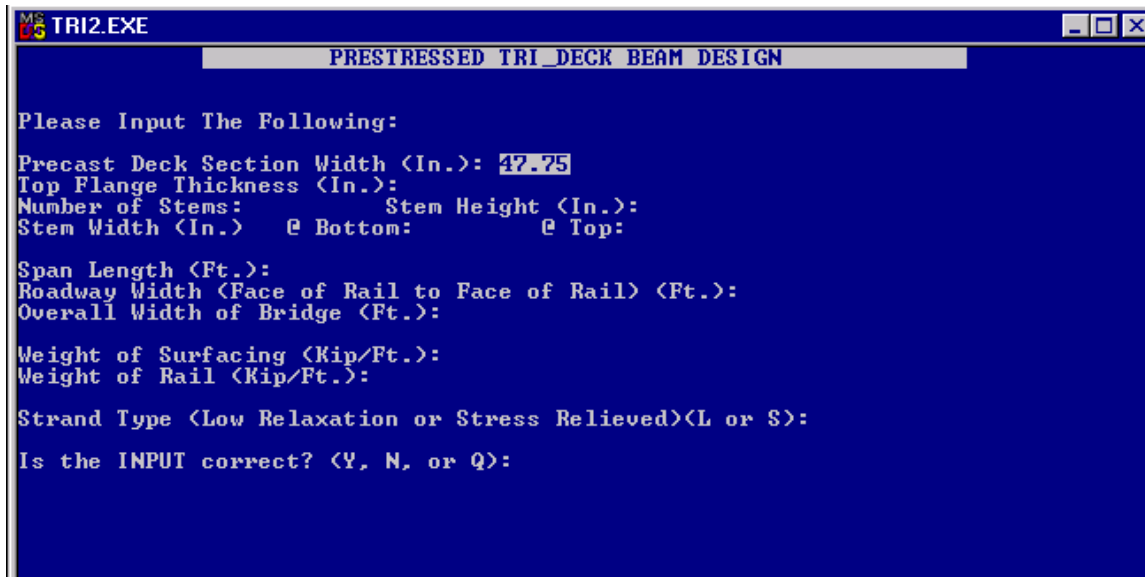
Verify data input.

**Title: Prestressed Tri-deck Beam Design****Location:** Bridge Programs\Superstructure\Tri Deck**Description:**

Given section dimensions and bridge properties, the program designs a beam section to meet the requirements.

**Input:**

Section dimensions, bridge dimensions, and future wearing surface and rail weight.



```
MS-DOS TRI2.EXE
PRESTRESSED TRI_DECK BEAM DESIGN

Please Input The Following:
Prest Deck Section Width (In.): 47.75
Top Flange Thickness (In.):
Number of Stems:      Stem Height (In.):
Stem Width (In.)   @ Bottom:      @ Top:

Span Length (Ft.):
Roadway Width (Face of Rail to Face of Rail) (Ft.):
Overall Width of Bridge (Ft.):

Weight of Surfacing (Kip/Ft.):
Weight of Rail (Kip/Ft.):

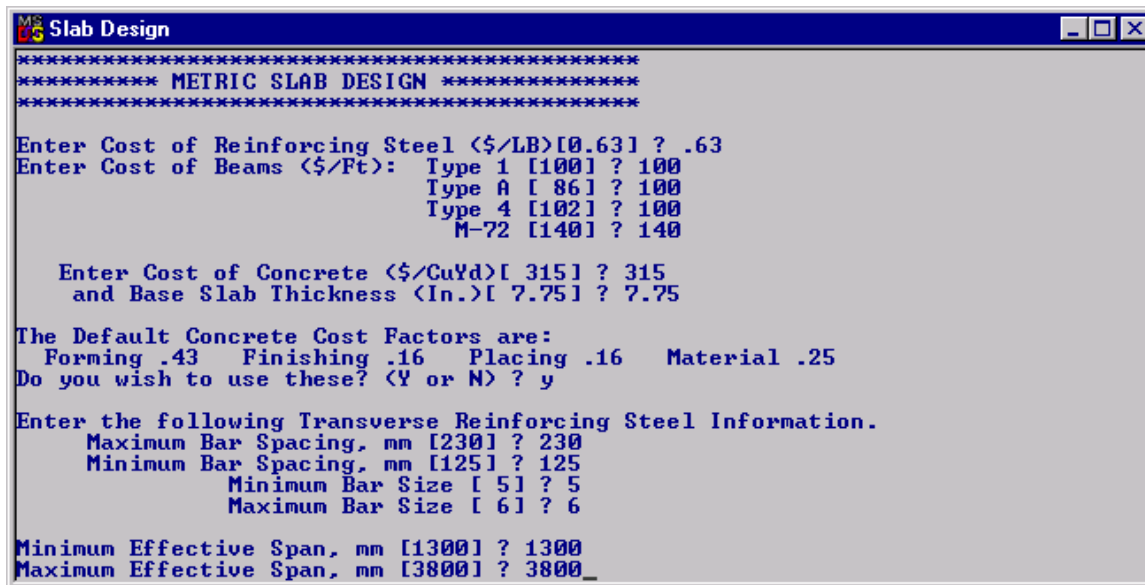
Strand Type (Low Relaxation or Stress Relieved)(L or S):
Is the INPUT correct? (Y, N, or Q):
```

**Output:**

The program provides section properties for the section (moment of inertia, section modulus, centroid location), dead and live load moments, dead load deflections, and allowable and actual stresses at transfer and in final configuration. It also generates a possible strand pattern.

**Title: Slab Design****Location:** Bridge Programs\Superstructure\Slab Design**Input:**

Enter the appropriate costs. Note that the default costs are not current.



```
MS-DOS Slab Design
*****
***** METRIC SLAB DESIGN *****
*****
Enter Cost of Reinforcing Steel (<$/LB>)[0.63] ? .63
Enter Cost of Beams (<$/Ft>):  Type 1 [100] ? 100
                             Type A [ 86] ? 100
                             Type 4 [102] ? 100
                             M-72 [140] ? 140

Enter Cost of Concrete (<$/CuYd>)[ 315] ? 315
and Base Slab Thickness (In.)[ 7.75] ? 7.75

The Default Concrete Cost Factors are:
Forming .43  Finishing .16  Placing .16  Material .25
Do you wish to use these? (Y or N) ? y

Enter the following Transverse Reinforcing Steel Information.
Maximum Bar Spacing, mm [230] ? 230
Minimum Bar Spacing, mm [125] ? 125
Minimum Bar Size [ 5] ? 5
Maximum Bar Size [ 6] ? 6

Minimum Effective Span, mm [1300] ? 1300
Maximum Effective Span, mm [3800] ? 3800_
```

**Output:**

The program generates a table of possible deck thicknesses at different effective slab spans and reinforcement patterns for each. It generates a similar list for each type of prestressed beam listed in the screen view. It offers an option to filter the list to show only the most economical sections. Note that the unfiltered list runs twenty-six pages.

**Title: Distribution of Flexural Steel**

**Units:** Available in both Metric and English Units

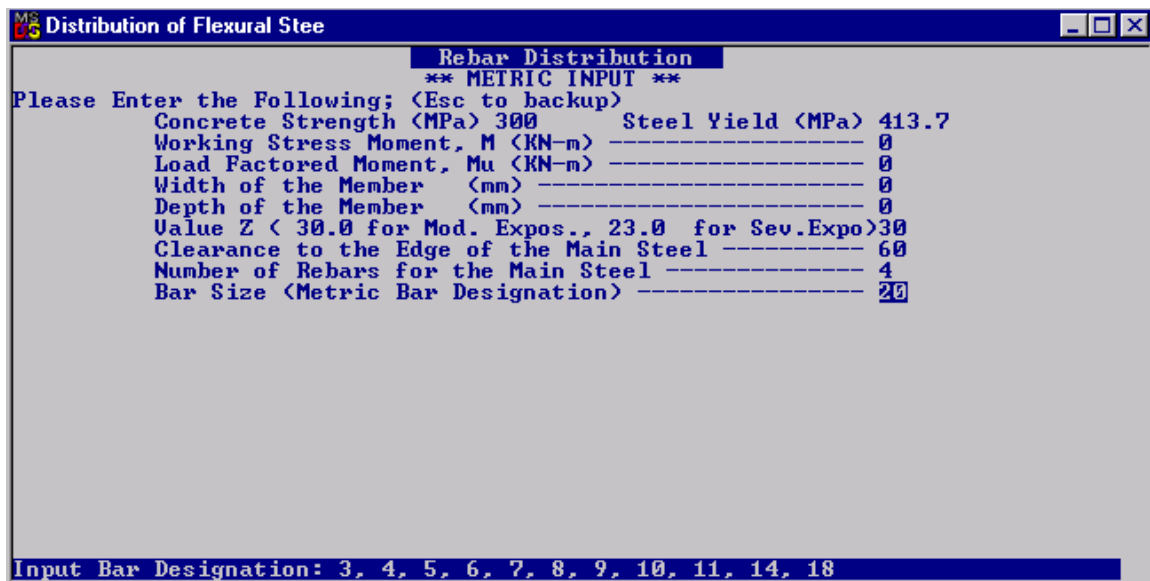
**Location:** Metric Version - Bridge Programs\Metric\Distribution of Flexural Steel  
English Version - Bridge Programs\Substructure\Distribution of Flexural Steel

**Description:**

The program calculates the primary steel required for a rectangular concrete flexural member. Assume the initial flexural steel and check against the design moments for the section. Program calculates the cracking moment, area of steel required by load factored moment and tensile stresses in mild steel. Iteration determines the final area of flexural steel.

The structural analysis is based on the requirements of the AASHTO **Standard Specifications for Highway Bridges, Sixteenth Edition** with '97 Interims (Section 8.16.8.4).

The structural analysis also applies to the requirements of the AASHTO **LRFD Bridge Design Specifications**, Second Edition 1998 with '01 Interims (Section 5.7.3.2).

**Input Screen:****Input Definitions:**

Required information: Design moments for the analyzed section. The working stress moment equals the Service Limit State moment.

Concrete Strength,  $F'_c =$  \_\_\_\_\_ MPa or ksi (Default 21 MPa or 3 ksi)

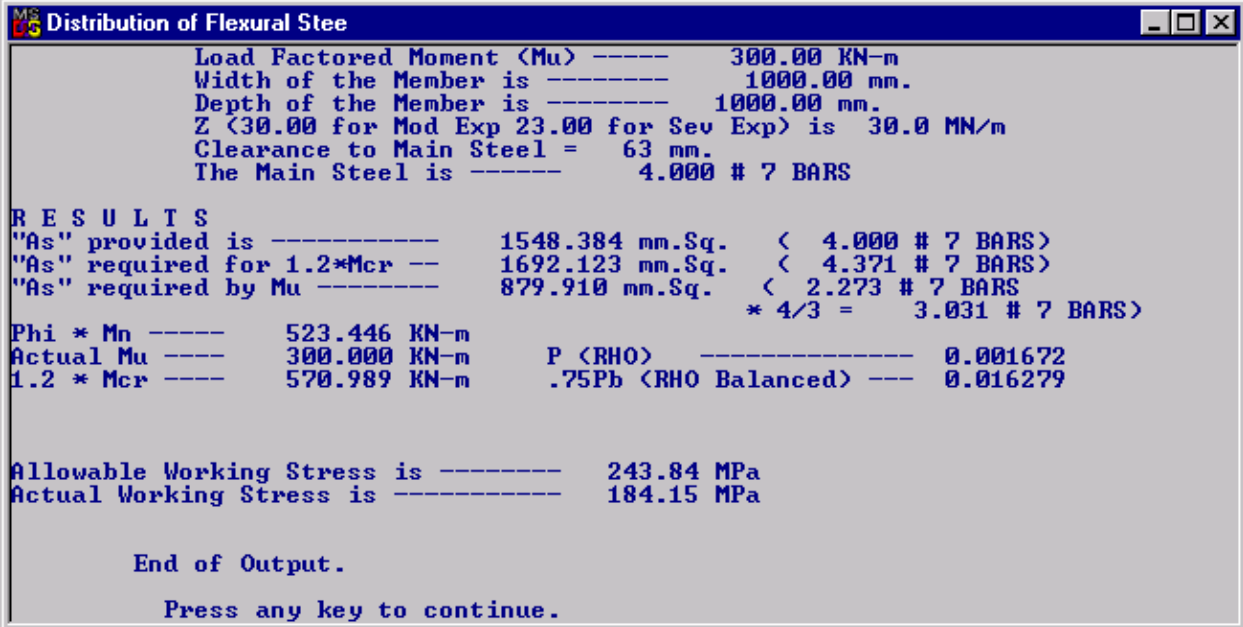
Steel Yield,  $F_y =$  \_\_\_\_\_ MPa or ksi (Default 413.7 MPa or 60 ksi)

Working Stress Moment,  $M =$  \_\_\_\_\_ kN-m or ft-kips  
Load Factored Moment,  $Mu =$  \_\_\_\_\_ kN-m or ft-kips  
Width of Member,  $W =$  \_\_\_\_\_ mm or inches  
Depth of Member,  $D2 =$  \_\_\_\_\_ mm or inches  
Crack Width Parameter,  $Z =$  \_\_\_\_\_ 30 kN/mm or 170 kips/in for moderate exposure (Default)  
23 kN/mm or 130 kips/in for extreme exposure  
Clearance to the Edge of the Main Steel,  $D3 =$  \_\_\_\_\_ mm or inches (Default 60 mm or 2.5 in)  
Number of Rebars for the Main Steel,  $N =$  \_\_\_\_\_ bar fractions acceptable (Default 4 bars)  
Enter the Bar Size,  $S =$  \_\_\_\_\_ (Default #6) (Allowed bar designations are English only)

### Output Screen:

Error messages that require parameter modifications and rerun of the program due to failure of code requirements:

!!! The Area of Steel is Insufficient Check Section 8.17.1.1-2 AASHTO  
!!! The Area of Steel is Insufficient for Mu.  
!!! The actual working stress is too high.



```
Distribution of Flexural Steel
Load Factored Moment <Mu> ----- 300.00 KN-m
Width of the Member is ----- 1000.00 mm.
Depth of the Member is ----- 1000.00 mm.
Z <30.00 for Mod Exp 23.00 for Sev Exp> is 30.0 MN/m
Clearance to Main Steel = 63 mm.
The Main Steel is ----- 4.000 # 7 BARS

R E S U L T S
"As" provided is ----- 1548.384 mm.Sq. < 4.000 # 7 BARS>
"As" required for 1.2*Mcr -- 1692.123 mm.Sq. < 4.371 # 7 BARS>
"As" required by Mu ----- 879.910 mm.Sq. < 2.273 # 7 BARS
* 4/3 = 3.031 # 7 BARS>
Phi * Mn ----- 523.446 KN-m
Actual Mu ----- 300.000 KN-m P <RHO> ----- 0.001672
1.2 * Mcr ----- 570.989 KN-m .75Pb <RHO Balanced> --- 0.016279

Allowable Working Stress is ----- 243.84 MPa
Actual Working Stress is ----- 184.15 MPa

End of Output.
Press any key to continue.
```

### Verification of Results:

Check the input values. Perform separate hand calculations to determine the required area of steel.

**Title: Reactions at End and Intermediate Bents****Location:** Bridge Programs\Metric\Reactions at End and Intermediate Bents**Description:**

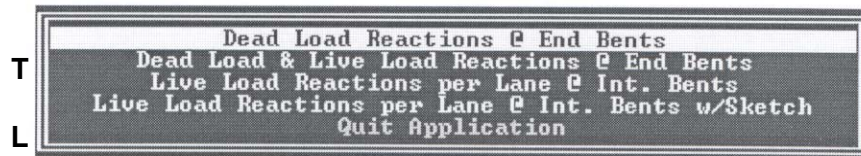
The program will calculate Dead Load reactions at the girder ends of a simple-span, prestressed girder bridge. It also has a routine that calculates LL reactions per lane at end or intermediate bents. The Program has four options: 1) DL Reactions at End Bents, 2) Combined DL and LL reactions per lane at end bents, 3) LL reactions per lane at intermediate bents, and 4) LL reactions per lane at intermediate bents with sketches.

**Limitations:**

- HS-20 loading, AASHTO Standard Specifications, 16<sup>th</sup> Edition.
- For simple-span designs only.
- The DL routine does not handle steel beams.

**Input:**

Need basic dimensions of spans, roadway width and backwall for dead load reactions. Input is self-explanatory from the menu. Program provides a toggle for concrete barrier rail or T101 rail.

**Verification of Results:**

Verify that the correct input values were used.

*Note: Dead Load Reactions output is kN/Girder for interior and exterior girders at the top of the cap. Cap weight is not included in the output.*

*Live Load Reactions output is kN/Lane at the end reaction. Sketches show if truck or lane loading controls and placement of the loads. End reactions need to split out to the individual girders as specified in Section 3 of the Standard Specifications.*



**Title: Round Column Design Using Strength Methods****Description:**

Summary: The program calculates the maximum moment the column cross section can resist while supporting the design axial load,  $P_u$ .

Explanation: Maximum concrete strain is assumed to be 0.003. Reinforcing bar strain and concrete compression block size are varied (constrained by proportional strain relationships) until the total axial resistance of the concrete and steel together,  $P_n$ , reduced for column understrength, is equal to  $P_u$ . The understrength is calculated as  $\phi P_n$ . From these same bar and concrete forces, the moment resistance,  $M_n$ , is also calculated along with the understrength moment resistance,  $\phi M_n$ .

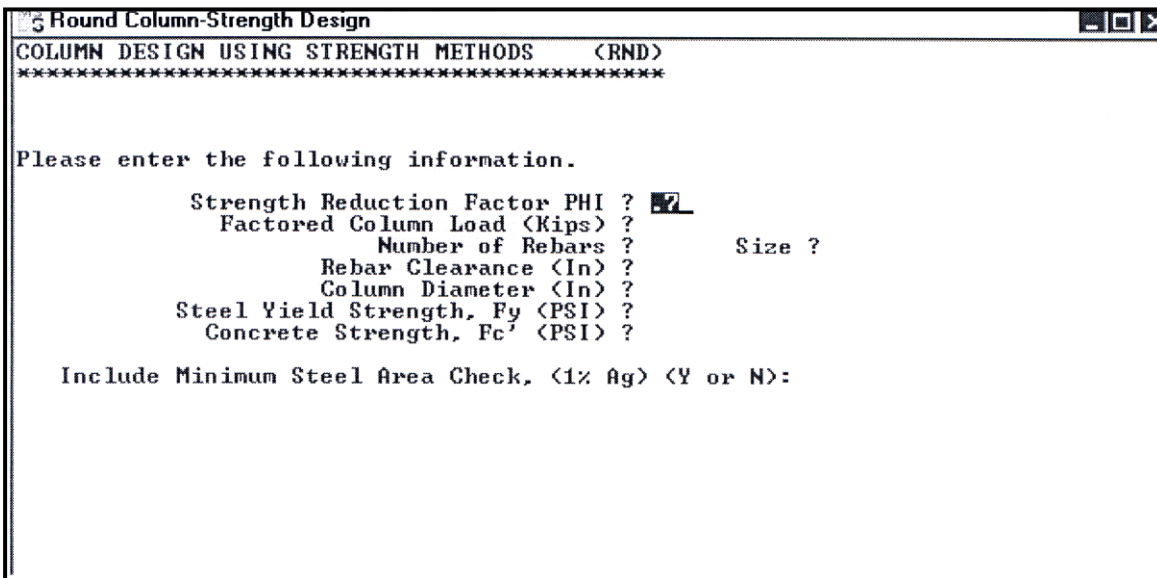
The routine calculates "Balanced Loading" by setting the bar strain at the extreme tensile side equal to the yield strain. The resulting axial and moment resistance is output as  $P(b)$  and  $M(b)$ , respectively.  $M_b$  is the maximum nominal moment resistance of the section. The corresponding axial load,  $P_u = \phi P_b$ , is therefore the optimum loading that gives maximum moment resistance. More or less applied  $P$  load will decrease moment capacity. Therefore, it is unconservative to assume extra dead or other axial loading for  $P_u < \phi P_b$  and conservative for  $P_u > \phi P_b$ .

The moment  $M_n$  is calculated using the moment arms from the bars and concrete compression block to the geometric centroid of the column section. For axial loads that put the total cross section in compression, the centroid of the concrete force is conservatively assumed to be at the geometric centroid. (No moment contribution from concrete.)

The program gives a warning when the nominal axial resistance,  $P_n$ , required to resist the applied load,  $P_u$ , exceeds the absolute compression limit of the section as calculated by Section 8.16.4.1.2 of the AASHTO Standard Specifications, 16<sup>th</sup> Edition. This limiting value used by the program assumes a spiral-reinforced column. For tie-reinforced columns, the limit is 94% of the spiral value output by the program. (For typical bridge designs, this limit will rarely be of concern because axial loadings must be low enough to leave some capacity to resist the high moments imposed by lateral loading. Therefore, for the usual types of loadings, the axial load on a suitable bridge column is much less than the limiting value given by the program.)

The program calculates the area of longitudinal steel reinforcing from the number of bars specified by the user. If the area of steel is less than the minimum allowed by Section 8.18.1.2 (i.e., 1% of the gross column cross sectional area ( $A_g$ )), the program gives the user the option of trying the same load on a column section where the specified reinforcement will be equal to 1% of  $A_g$ . The reasoning is that, if a column with minimum steel area can support the required loads, adding extra concrete cannot diminish capacity. Similarly, if having minimum steel in a column prevents excessive creep strain in the steel, that strain will not increase by adding extra perimeter concrete. The reduced column uses reduced bar clearance equal to: (Actual Clearance) X (Reduced Diameter/Actual Diameter).

No flag is output for reinforcing ratios exceeding the 8% specified by Section 8.18.1.1.

**Input:**

```
Round Column-Strength Design
COLUMN DESIGN USING STRENGTH METHODS  <RND>
*****
Please enter the following information.

      Strength Reduction Factor PHI ? 0.7
      Factored Column Load <Kips> ?
      Number of Rebars ?           Size ?
      Rebar Clearance <In> ?
      Column Diameter <In> ?
Steel Yield Strength, Fy <PSI> ?
Concrete Strength, Fc' <PSI> ?

Include Minimum Steel Area Check, <1% Ag> <Y or N>:
```

**Verification of Results:**

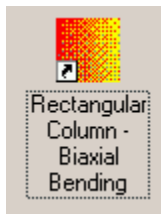
Verify that the correct input values were used.

**Title: Rectangular Column, Biaxial Bending – Strength Design****Location:** Bridge Programs\Substructure**Description:**

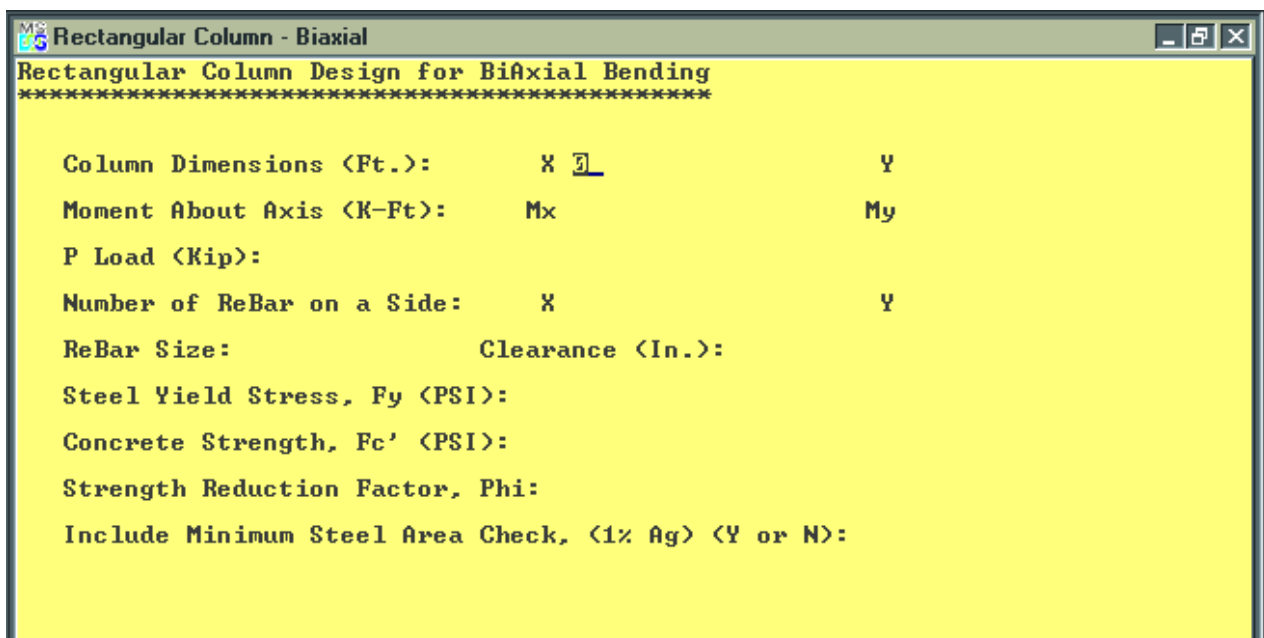
This program will investigate the adequacy of a square or rectangular column subject to uniaxial or biaxial loading based on the input provided by the user. In general, members subject to an axial load combined with bending are designed for the maximum moment that can accompany the axial load. This program calculates and compares the required eccentricity to the provided eccentricity. If the provided eccentricity is less than required, the program output will state to try a different design. Adjust the reinforcing steel and/or the column size until the provided eccentricity exceeds the required eccentricity.

**Input:**

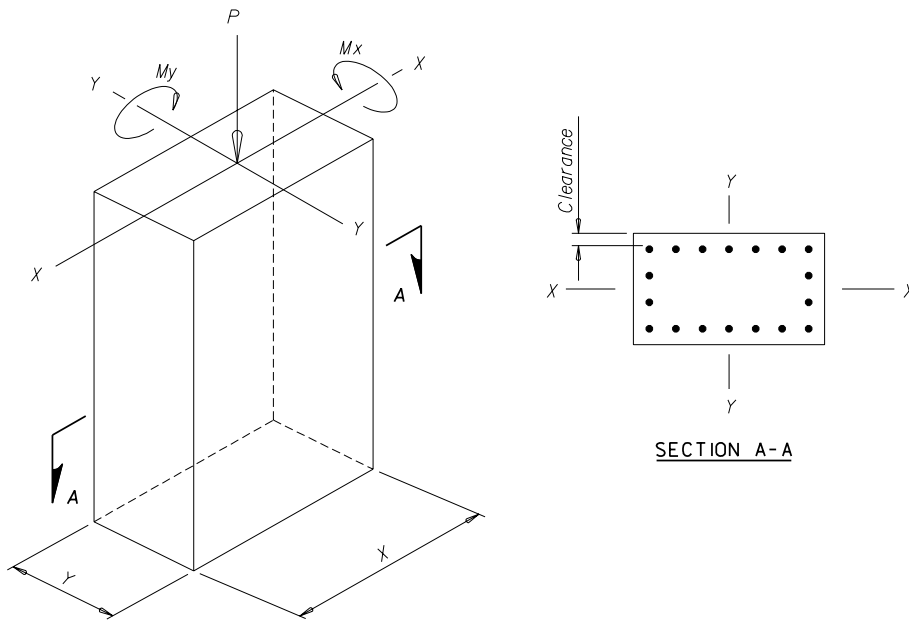
This program uses customary U.S units only. Begin by double clicking the icon.



The following screen will pop up.



For a description of the variables see the drawing below.



- **$M_x$**  = Calculated design moment about the X axis.
- **$M_y$**  = Calculated design moment about the Y axis.
- **$P$**  = Calculated design axial load.
- **Number of Rebar on a side:** Select a number of bars per face.  
In the drawing above,  $X=7$ ,  $Y=4$
- **Rebar size:** Enter the bar size, 7 thru 18.

*Note: Bridge Bureau policy states that the minimum number and size of bars per column is 8 #7 bars.*

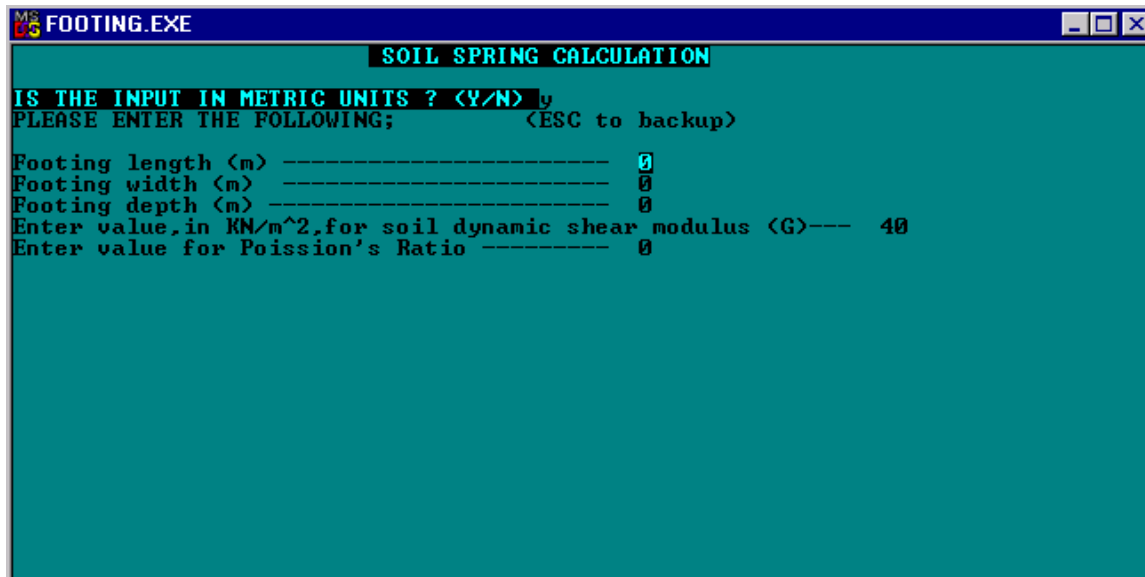
- **Clearance:** Distance from face of concrete surface to nearest edge of vertical steel. Default = 3.5 inches.
- **Steel Yield Stress,  $F_y$  (psi):** Default = 60,000 (Grade 60)
- **Concrete Strength,  $F'_c$  (psi):** Default = 3000 (Class DD concrete)
- **Strength Reduction Factor:** Default = 0.7
- **Include Minimum Steel Area Check, (1%  $A_g$ ) (Y or N):** Default = Y.

### Verification of Results:

Verify that the correct input values were used.

**Title: Soil Spring and Footing Stiffness****Location:** Bridge Programs\Other\footing.exe**Description:**

Calculates equivalent radii for spread footings and for footings on piles, shape factors ( $\alpha$ ) and embedment factors ( $\beta$ ) and footing stiffnesses about each of three axes for use in seismic design. (Method based on *Seismic Design of Highway Bridge Foundations*, FHWA Report Number FHWA/RD-86/101, Ignatius Po Lam and Geoffrey R. Martin, June 1986.)

**Input:**

```
MS-DOS FOOTING.EXE
SOIL SPRING CALCULATION
IS THE INPUT IN METRIC UNITS ? <Y/N>
PLEASE ENTER THE FOLLOWING;          <ESC to backup>
Footing length <m> ----- 0
Footing width <m> ----- 0
Footing depth <m> ----- 0
Enter value, in KN/m^2, for soil dynamic shear modulus <G>--- 40
Enter value for Poission's Ratio ----- 0
```

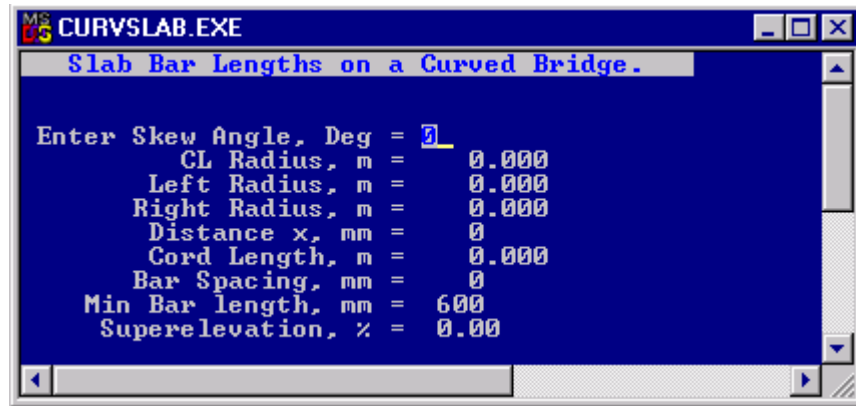
**Verification:**

Refer to the text to analyze output.

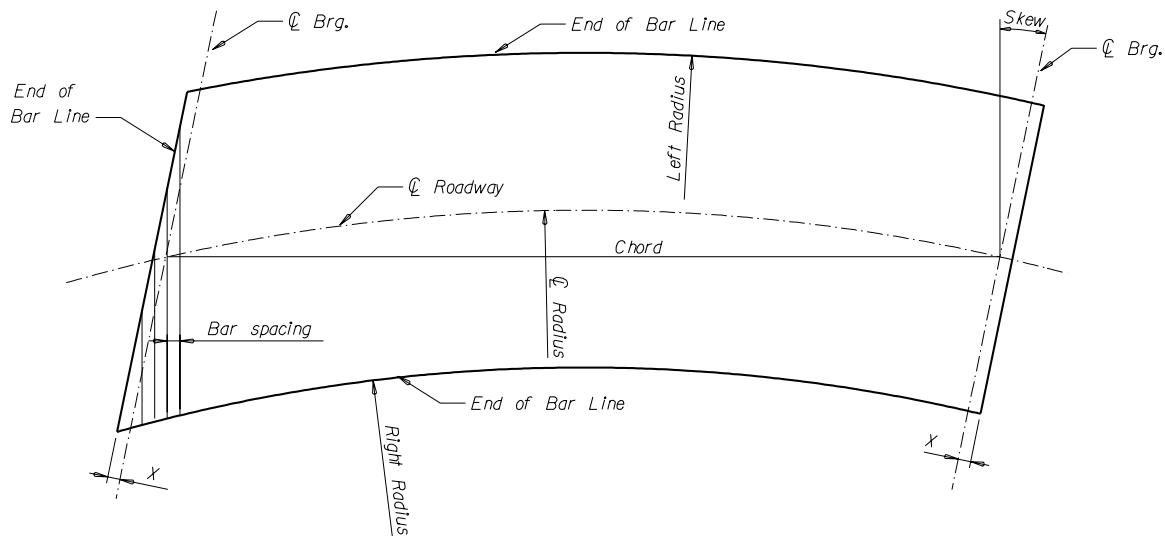
**Title: Slab Bar Lengths on a Curved Bridge**

**Location:** Bridge Programs\Metric\Curved Slab Bar Lengths

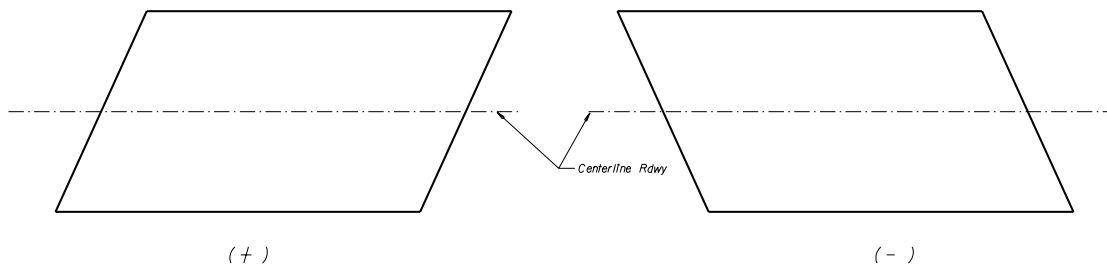
**Input:**



**Graphical Description:**



**Skew Sign Convention:**



**Input Definitions:**

Skew Angle: Measured from a perpendicular reference line to the bridge long chord generally unless there is an expansion joint in the slab (degrees). (See Skew Sign Convention)

Centerline Rdwy Radius: This program works for both right and left hand curves (meters).

Left Radius. This is the radius on the left side of the bridge that defines the end of the transverse slab bars at the edge of slab (meters). (0.075 meters from edge of slab)

Right Radius: Same as above except on the right side of the structure (meters).

Distance x: This point establishes first and last transverse slab bar. This distance is the perpendicular distance from centerline bearing to the end of the slab, and then subtract 40 mm clearance to establish the first bar position (mm). (Note: If x is not the same on both ends, run the program twice inputting first one side and then the next side.)

Chord Length: Chord length of continuous slab portion for which rebar lengths are being calculated (meters).

Bar Spacing: Centerline bar spacing of transverse bars measured perpendicular to bar layout (mm).

Minimum Bar Length: Default is 610 mm.

Superelevation: (%)

**Output:**

- Input values are displayed for verification.
- Bar lengths are given in skewed section of bridge on both ends.
- The units on the distance from centerline bearing of first bridge is shown as meters; however, the program prints the output in mm.
- Additionally, the program gives bar length difference between center of bridge and last full length bar before the skew at both ends. This is helpful to see if different bar lengths are needed.

**Title: Skewed Slab Bar Lengths**

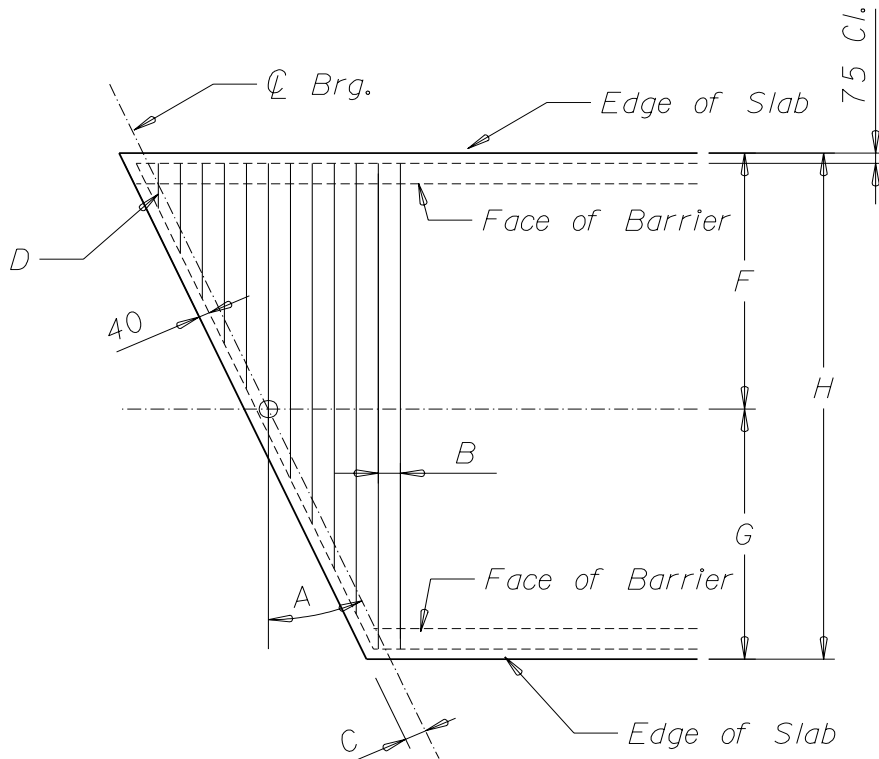
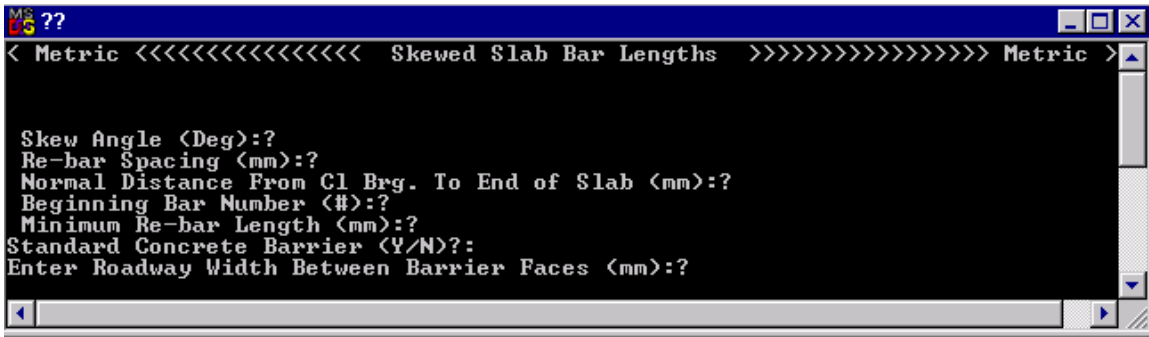
**Location:** Bridge Programs\Metric\Bar Lengths (Skewed Slab)

**Description:**

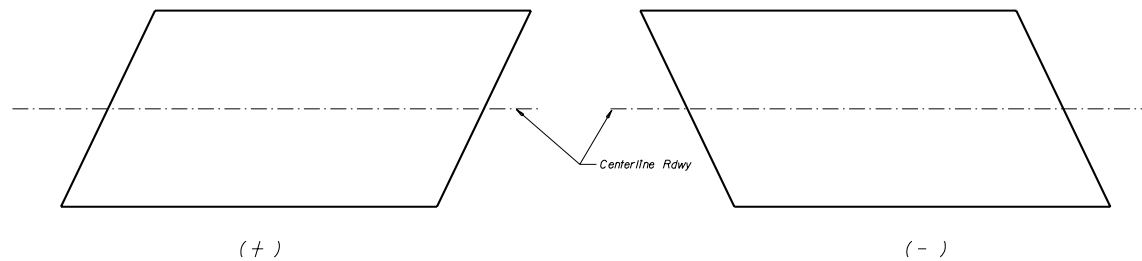
This program calculates the number and length of the variable length transverse slab bars in the skewed section of a tangent bridge.

**Input Screen:**

The program will look similar to the screen shown below. (The program prompts for input one line at a time.)





**Skew Sign Convention:****Input Definitions:**

- A - Skew Angle: (See diagram for sign convention)
- B - Rebar Spacing: Centerline spacing of transverse reinforcement measured along chord
- C - Perpendicular distance from centerline bearing to end of slab (see diagram)
- D - Beginning Re-bar Number: Use bar numbers to match Bill of Reinforcing Steel
- E - Minimum Re-bar Length: (Standard is 610 mm)

Standard Concrete Barrier?

If no, the program prompts for dimensions F and G.

If yes, input H.

**Hints:**

A peculiarity of the program is that, when standard concrete barrier rail is specified, the program only allows for input H. This does not allow for offset centerline of roadway. So when the bridge has an offset alignment with concrete barrier rail, input N when prompted for Standard Concrete Barrier. Then input dimensions F and G (centerline roadway to edge of slab).

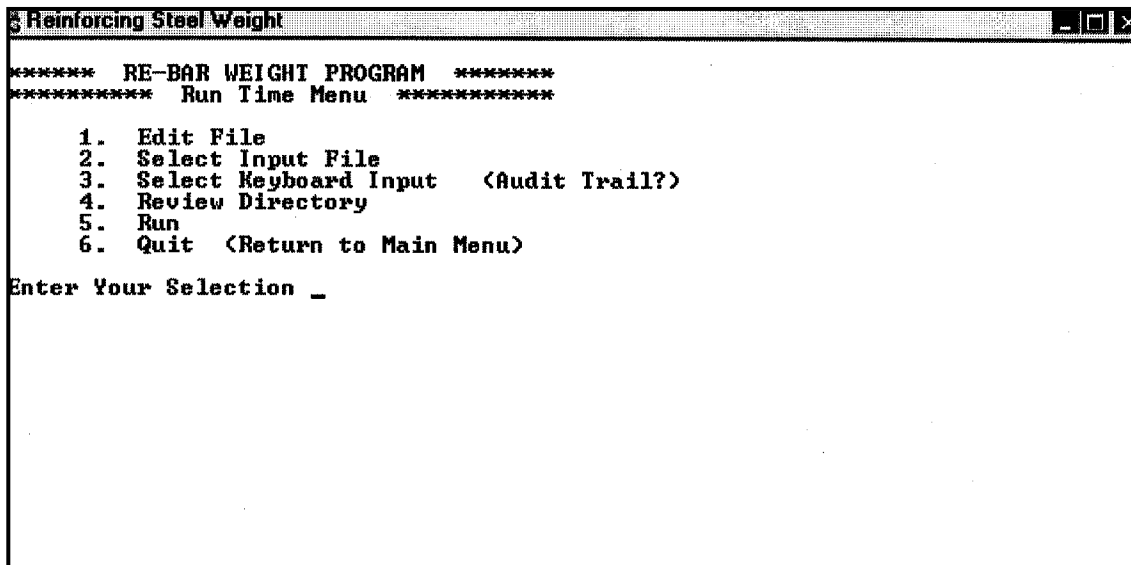
**Output:**

Input values are displayed for verification. For the Standard Barrier Rail, the program calculates the dimension from centerline roadway to the slab edges based on the assumption that the centerline of roadway is centered between barriers.

The program calculates the lengths of each bar in the skewed section and assigns a bar number. In addition, the bar number and length of the rebar passing through the centerline of bearing is noted.

**Title: Reinforcing Steel Weight****Location:** Bridge Programs\Metric\Reinforcing Steel Weight**Description:**

The program calculates and totals reinforcing steel weight for each bar size. Input the bar callout, the number of bars, the bar size and the length of the bar from the bill of steel. The program calculates the weight for each bar callout and the total weight for each bar size for Plain bars, Epoxy bars, Seismic bars or Epoxy-Seismic bars. This information is then entered into the Quantity Sheet and is used for estimating project costs by MDT and by the Contractors bidding the project.

**Input:**A screenshot of a DOS-style window titled "Reinforcing Steel Weight". The window contains a menu with the following text:

```
***** RE-BAR WEIGHT PROGRAM *****
***** Run Time Menu *****

1. Edit File
2. Select Input File
3. Select Keyboard Input <Audit Trail?>
4. Review Directory
5. Run
6. Quit <Return to Main Menu>

Enter Your Selection _
```

**Directions:**

To start a new rebar run, select number 2 from the menu. Input filename consisting of the Control No. followed by the description and a .txt extension. Example: 1515Bt1.txt. The TXT file is usually saved in the same place as the program; in this case, it will be saved under the Bridge Programs in the metric folder.

Pick number 3 to input the data. On the first part of the bar mark, put the alphabetic coding for the bar (like C for a cap bar or BW for a backwall bar). Press the enter key to move to the next line. Enter the first bar number of the bar series, and press enter to move to the next line. Enter the last consecutive number of the bar series. Press the enter key to move to the next line.

*Note: If there is a bar that is not in a consecutive series or if a bar mark is different from the rest of the bars, code the first and last bar the same.*

The program will ask if the bars are epoxy coated or not; press Y if they are or press N if they are not. Follow the same process for seismic bars.

The next screen will ask for the bar size and the number of bars and the total length of the bar in millimeters. The program will go to the next bar in the run and ask for the bar size, number of bars and the total length of bar. Once the input for all bars in a series is complete, the program will return to the first screen and ask for a new alphabetic bar mark, and it will follow the same input above for each bar mark in the bill of reinforcing.

Once all of the bars are input, a summary screen will show the bar sizes and the total weight for that bar size for Plain, Epoxy, Seismic and Epoxy-Seismic.

Review the data prior to printing. If the information appears correct, print as indicated below. If corrections are needed, see directions below.

In the lower right corner of the next screen, the user will be prompted to print the output. Press Y if yes or press N if no.

The program will return to the beginning screen ready for the next bill of reinforcing to be input or, if the user is ready to quit the program, then select number 4.

If the data needs editing, go to notebook or other text editor. Open the TXT file that was created and make the changes in the TXT file and resave it. Go back into the Reinforcing steel weight program and select number 1. Type in the name of the file for which the changes were made to select number 3. It will give an overview of the bars. When done reviewing, press any key to move to the print screen. Press Y if yes or press N if no.

The program will return to the beginning screen ready for the next bill of reinforcing to be input or, if the user is ready to quite the program, then select number 4.

### **Verification of Results:**

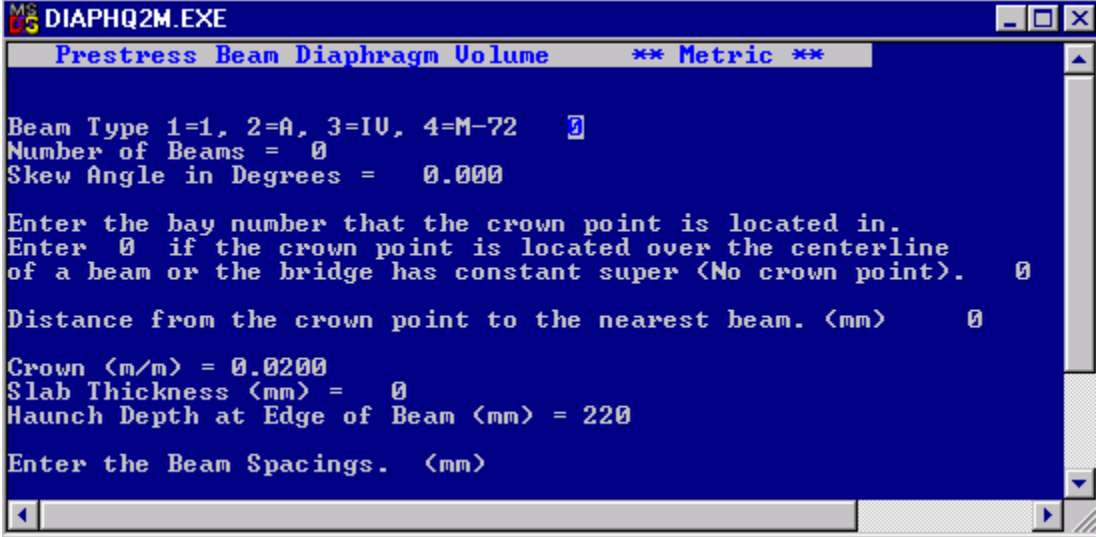
Verify that the correct input values were used by comparing the output sheets to each Bill of Reinforcing for the plans. Correct and rerun if needed. See Directions above.

**Title: Prestressed Beam Diaphragm Volume****Location:** Bridge Programs\Metric\Diaphq2m**Description:**

This program calculates the concrete diaphragm volume for standard MDT girder shapes.

**Input Screen:**

The program will look similar to the screen shown below. (The program prompts for input one line at a time.)



```
MS-DOS DIAPHQ2M.EXE
Prestress Beam Diaphragm Volume    ** Metric **

Beam Type 1=1, 2=A, 3=IU, 4=M-72
Number of Beams = 0
Skew Angle in Degrees = 0.000

Enter the bay number that the crown point is located in.
Enter 0 if the crown point is located over the centerline
of a beam or the bridge has constant super (No crown point). 0

Distance from the crown point to the nearest beam. (mm) 0

Crown (m/m) = 0.0200
Slab Thickness (mm) = 0
Haunch Depth at Edge of Beam (mm) = 220

Enter the Beam Spacings. (mm)
```

**Input Definitions:**

The input for this program is explained in the above input screen.

**Output:**

Input values are displayed for verification.

The program computes Class DD concrete volumes in both end diaphragms and intermediate diaphragms. The volume of DD computed is to the bottom of the fillets shown on the SL-5 and SL-6 standard slab drawings. The program also calculates Class Special Deck concrete quantities above the bottom of the fillet and below the bottom of the slab. All quantities are for one line of diaphragms; therefore, the output will need to be multiplied by the number of diaphragm lines for each diaphragm type.

**Title: End Bent Quantities**

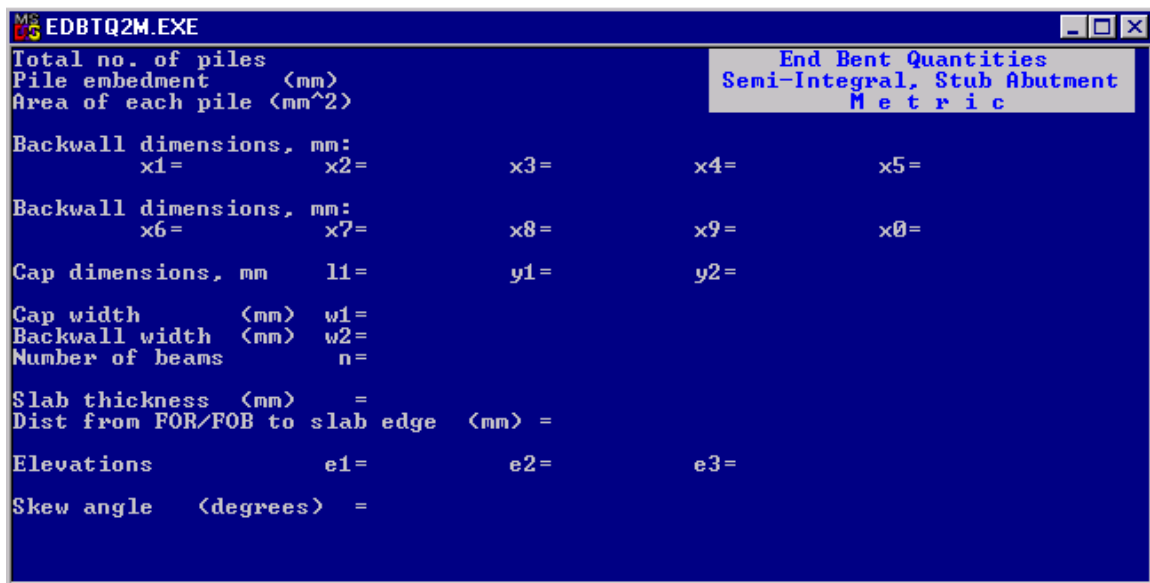
**Location:** Bridge Programs\Quantities\End Bent Quantities for English or Bridge Programs\Quantities\Metric\End Bent Quantities for Metric

**Description:**

This program calculates the volume of Special Deck and DD concrete in a typical stub end bent. Input the geometry of the end bent with elevations.

**Input:**

Below is what the program will look like on the screen.



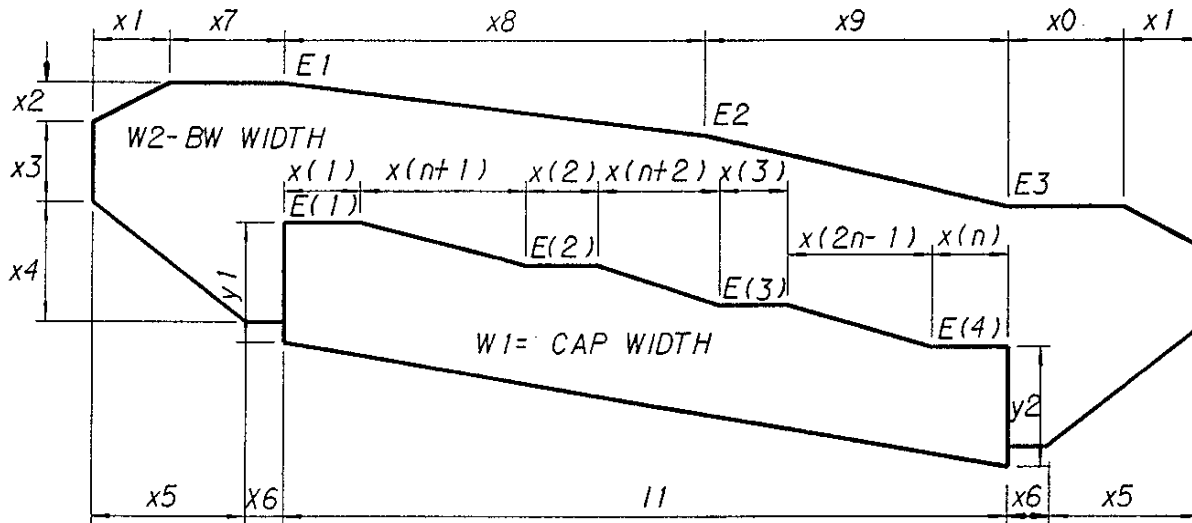
Use the following Input sheets for ease of entry.

**HINTS:** For dimensions X6 through X0, use dimensions figured along the Fill face of the backwall.

When calculating volumes for an end bent using steel beams, enter any type of beam. No deductions are made for prestressed or steel beams.

Add the Special deck volume and paving notch volume back in for end bents used with Bulb T structures.

Add the paving notch volume to the Special Deck volume for end bents that do not have a paving notch.



Total no. of piles \_\_\_\_\_  
 Pile embedment (Default 1) \_\_\_\_\_ ft  
 Area of each pile \_\_\_\_\_ ft<sup>2</sup>

Backwall dimensions (x1-x5):  
 (Default 2)  
 x1 = \_\_\_\_\_ ft  
 x2 = \_\_\_\_\_ ft  
 x3 = \_\_\_\_\_ ft  
 x4 = \_\_\_\_\_ ft  
 x5 = \_\_\_\_\_ ft

Backwall dimensions (x6-x0):  
 (Default 1)  
 x6 = \_\_\_\_\_ ft  
 x7 = \_\_\_\_\_ ft  
 x8 = \_\_\_\_\_ ft  
 x9 = \_\_\_\_\_ ft  
 x0 = \_\_\_\_\_ ft

Cap dimensions:  
 l1 = \_\_\_\_\_ ft  
 y1 = \_\_\_\_\_ ft  
 y2 = \_\_\_\_\_ ft

Cap width (Default 3) w1 = \_\_\_\_\_ ft  
 Backwall width (Default 1.5833) w2 = \_\_\_\_\_ ft  
 Number of beams n = \_\_\_\_\_

## Stub Endbent Quantities

Page 2 of 2

Slab thickness = \_\_\_\_\_ in  
 Dist from FOR/FOB to slab edge = \_\_\_\_\_ ft (default = 1.1667)

Elevations  
 e1 = \_\_\_\_\_  
 e2 = \_\_\_\_\_  
 e3 = \_\_\_\_\_

Skew angle = \_\_\_\_\_ degrees

Beam type(1=1, 2=A, 3=IV, 4=M-72, 5=BT-72, 6=OTHER) = \_\_\_\_\_

For other types of beams:

Slab thickness at edge of slab = \_\_\_\_\_ in  
 Slab thickness at edge of ext. beam = \_\_\_\_\_ in  
 Dist from ext. beam edge to slab edge = \_\_\_\_\_ ft

Horizontal length & elevation of one beam seat:

x(1) = _____	ft	e(1) = _____
x(2) = _____	ft	e(2) = _____
x(3) = _____	ft	e(3) = _____
x(4) = _____	ft	e(4) = _____
x(5) = _____	ft	e(5) = _____
x(6) = _____	ft	e(6) = _____
x(7) = _____	ft	e(7) = _____
x(8) = _____	ft	e(8) = _____
x(9) = _____	ft	e(9) = _____
x(10) = _____	ft	e(10) = _____

Slope distance between beams:

x(n+1) = _____	ft
x(n+2) = _____	ft
x(n+3) = _____	ft
x(n+4) = _____	ft
x(n+5) = _____	ft
x(n+6) = _____	ft
x(n+7) = _____	ft
x(n+8) = _____	ft
x(n+9) = _____	ft

**Verification of Results:**

Verify that the correct input values were used.

**Title: Area Centroid Moment of Inertia****Location:** Bridge Programs\Geometry**Description:**

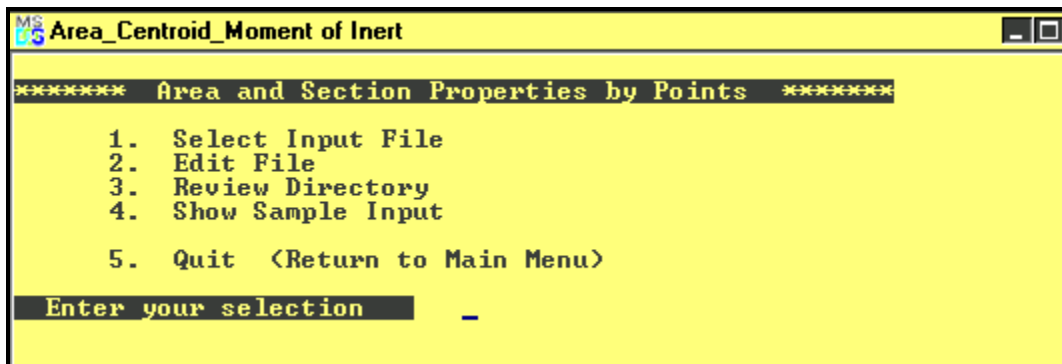
Program calculates the location of the centroid, moment of inertia, distance from neutral axis to extreme fiber, section moduli and radius of gyration about the x-axis and y-axis.

**Input:**

Start by creating an input file in a file editor program such as Notepad. See the example input file below. Save the file to C:\Programs\Geometry.

```
***** Example Input File *****
9.5 -1.375      ' Enter one point per line. Coordinate values may be
9.5,0          ' comma seperated or space seperated.
.3125 0
.3125 100      ' Comments may be placed on any line. Comments are
8.35, 100     ' ignored.
8.35 101.125
sy            ' More then one set of data may be in one file. Seperate
next         ' the date with the word NEXT.
.3125 16.985
5.9875 16.985
5.9875 17.925
sy           ' SX and SY indicate that the section is symmetric about
sx           ' the X or Y axis. The additional points are generated.
*****
```

Save the file to C:\Programs\Geometry where the program will be able to read it.



Select the input file option. Type in the name of the input file created earlier.

**Verification of Results:**

Verify that the correct input values were used.



### 25.3 EXTERNAL PROGRAMS

See Figure 25.3A.

	Name	Location
1.	AISIBEAM	Bridge Programs\Superstructure\AISIBEAM
2.	AISIsplce	Available on disk
3.	BRASS-GIRDER(LRFD) <sup>™</sup>	Programs\BRASS LRFD
4.	BT Beam	Bridge Programs\Substructure\BT Beam
5.	CONSPAN LA <sup>™</sup>	Bridge Programs\Superstructure\CONSPAN LA
6.	Florida Pier	Internet Download
7.	GEOPAK Bridge	Accessed through Microstation
8.	LPILE Plus	Bridge Programs\Other\LPILE Plus
9.	Mathcad®	Bridge Programs\Other\Mathcad
10.	MERLIN-DASH	Bridge Programs\Superstructure\Merlin Dash
11.	Opis	Desktop Icon
12.	Programmers File Editor (PFE)	Bridge Programs\Other\ Programmers File Editor
13.	Pontis	MDT Intranet
14.	SEISAB	Bridge Programs\Other\Seisab
15.	SIMON Systems	Available on disk
16.	System VANCK	Available on disk
17.	Virtis	Desktop Icon

### EXTERNAL COMPUTER PROGRAMS

Figure 25.3A

**Program Name: AISIBeam****Location:** Bridge Programs\Superstructure\AISIBeam**Description:**

The software is based on the Strength Design Method (Load Factor Design) of the AASHTO **Standard Specifications for Highway Bridges**, 16th Edition, 1996, including the 1997, 1998, and 1999 Interim Specifications. The software uses Group I LFD load combinations. The Software will perform a line-girder analysis for simple-span girder and rolled beam bridges.

**Inputs:**

The following items can be customized by the user:

- Welded plate girders with unstiffened webs
- Rolled beams with optional welded cover plates
- Composite and non-composite stringers
- Normal weight and lightweight concrete deck material
- AASHTO strength, deflection, fatigue and constructibility criteria
- HS (MS) type truck load or special loading with up to 15 axle loads
- Up to 25 alternative solutions for each design
- Supporting design calculations
- Customary U.S. units or SI (metric units)
- Alternative fatigue load of HS (MS) configuration
- Operating and Inventory Ratings

**Output:**

In the design mode, the software will iterate between a range of user-specified minimum and maximum cross section dimensions to find a minimum-weight solution. In the rating mode, the user can input the exact cross section properties, and the Software will then solve for both an Inventory and Operating Rating.

**Program Name: AISlsplice****Location:** Available on disk**Description:**

AISlsplice is a tool for the analysis and design of bolted field splices for straight, right, I-shaped, steel girders. The analysis and design process is based on the AASHTO **LRFD Bridge Design Specifications**, Second Edition, 1998, including the 1999 interim.

In the design mode, the software sizes and optimizes the splice plates and bolts. In the analysis mode, the software determines the adequacy of given splice plates and bolts. For both modes, performance ratios (load/resistance) for all splice components are determined.

**Program Name: BRASS-GIRDER(LRFD)<sup>™</sup>****Location:** Programs\BRASS LRFD**Description:**

BRASS-GIRDER(LRFD)<sup>™</sup> is a comprehensive system for the analysis of highway bridge girders. BRASS-GIRDER(LRFD)<sup>™</sup> utilizes finite element theory of analysis and current AASHTO Load Resistance Factor Design (LRFD) Specifications.

**Inputs:**

BRASS-GIRDER(LRFD)<sup>™</sup> uses Windows based Graphical User Interfaces (GUI) for data input. Input and output may be created using US Customary or SI Units.

System input is free format consisting of commands grouped logically to define the bridge structure, loads to be applied and the output desired.

Girder types may be simple span, continuous, hinged or cantilevered, with or without integral leg frame configuration. Girders may be constructed of steel, reinforced concrete or prestressed concrete (pre- and post-tensioned). Composite steel and composite prestressed concrete girders may be included. BRASS can analyze variable depth girders, such as tapered or parabolic. The user may specify (by name) predefined cross sections that are stored in the cross section library. The library contains nearly all AISC rolled wide flange shapes and most AASHTO standard shapes for prestressed concrete I-beams. Using a library utility program, the user may modify the geometry of the existing sections, add new sections or delete existing sections.

Stage construction may be modeled by respective cycles of the system for girder configuration and load application. Cycles are automatic if desired. The dead load of structure members is automatically calculated if desired. Additional distributed loads and point loads may be applied in groups and each group assigned to a specific construction stage. Distributed loads may be uniform or tapered and divided into sections to model sequential slab pours. Loads due to prestressing are calculated and applied internally. Live loads may be moving trucks or uniform lanes loads, which include the HL-93 vehicles described in the AASHTO LRFD Specifications. Impact may be user defined, as specified by AASHTO, or the user may reduce impact to model reduced speed limits.

**Outputs:**

The program computes moments, shears, axial forces, deflections and rotations caused by dead loads, live loads, settlements and temperature changes. These actions are utilized by various subroutines to analyze user-specified sections of the girder.

**Program Name: BT Beam****Location:** Bridge Programs\Substructure\BT Beam**Description:**

BT Beam is an analysis engine for the analysis of simple and continuous girders in accordance with the AASHTO **LRFD Bridge Design Specifications**.

**Inputs:**

The input is a description of the girder in terms of geometry, boundary conditions, loading and code-compliance requirements.

**Outputs:**

BT Beam reports influence lines for each live load and then computes factored and distributed loads. Finally, load envelopes are reported. Output is ASCII and delimited table for importing into a spreadsheet.

**Program Name: CONSPAN LA™****Location:** Bridge Programs\Superstructure\CONSPAN LA**Description:**

CONSPAN LA™ is a comprehensive program for the AASHTO LFD and LRFD design and analysis of simple-span and multiple-span bridges, constructed with prestressed precast concrete beams, in U.S. and SI units. CONSPAN LA™ incorporates both AASHTO Specifications into one interface. The user can design using one code, and simply toggle to the other code for quick and complete design comparisons. This feature makes the transition from AASHTO LFD Specifications to AASHTO LRFD Specifications simple and efficient.

Simple-span static analysis is performed for dead loads resisted by the precast sections. Continuous static analysis is performed for dead loads acting upon the composite structure. A continuous moving load analysis is performed for the live load.

**Inputs:**

CONSPAN LA™ makes the entry of project data convenient with a system of tab screens, dialog boxes, graphical button, menus and wizards. Designs are completed with CONSPAN LA's™ automated features.

**Outputs:**

CONSPAN LA™ presents analysis results in a variety of easy to view formats, from a one page design summary to comprehensive project reports. Analysis results and graphical sketches can be exported to spreadsheets and DXF Formats.

During individual beam designs, various design parameters such as distribution factors, impact/dynamic allowance factors and allowable stresses are established. The strand and debonding/shielding patterns can be automatically generated by CONSPAN LA™ or specified by the user. Debonding constraints limiting the number of debonded strands can also be user-specified. Service load stress envelopes, generated by combining the results of the analysis, are checked against allowable limits. Factored positive moments and shears are checked against the ultimate strength capacity of the effective section. Mild reinforcement in the deck, at the piers, is computed for factored negative and positive moments. Many other code criteria such as cracking moments, horizontal shear, stresses at limit states, etc., are also automatically checked.

**Program Name: Florida Pier****Location:** Internet Download**Description:**

The Florida Pier analysis program is a nonlinear finite element analysis program designed for analyzing bridge pier structures composed of nonlinear pier columns and cap supported on a linear pile cap and nonlinear piles/shafts with nonlinear soil. This analysis program couples nonlinear structural finite element analysis with nonlinear static soil models for axial, lateral and torsional soil behavior to provide a robust system of analysis for coupled bridge pier structures and foundation systems.

**Inputs:**

Florida Pier performs the generation of the finite element model internally given the geometric definition of the structure and foundation system as input graphically by the designer. This allows the engineer to work directly with the design parameters and lessens the bookkeeping necessary to create and interpret a model.

**Outputs:**

Florida Pier contains an analysis program, FLPIER, that is coupled with a graphical pre-processor FLPIER\_GEN and post-processor FLPIER\_PLOT. These programs allow the user of Florida Pier to view the structure while generating the model as well as view the resulting deflections, bi-axial and uni-axial interaction diagrams and internal forces in a graphical environment.

**Program Name: GEOPAK Bridge**

**Location:** Accessed through MicroStation

**Description:**

GEOPAK Bridge represents technology for object-oriented bridge design and bridge design project management, integrated with civil engineering. Offering a toolkit of bridge modeling options, the software can be used to model most types of bridge structures, from simple to complex.

GEOPAK Bridge is fully integrated with MicroStation and the GEOPAK Civil Engineering Suite software, enabling interactive modeling in a familiar environment while ensuring the necessary integration with other transportation design disciplines.

**Inputs:**

The software includes a Bridge Project Explorer that gives access to bridge components and their properties, stored in a single bridge project database.

The software includes a set of sample report templates for startup.

**Outputs:**

The software reports any failures and automatically adjusts the location of bearings and beam seats to comply with the user's haunch requirements.

A Reports Manager gives flexibility to custom design reports that meet the requirements of a project or organization.

The user can test deflection conditions using five different deflection methods. GEOPAK Bridge models its solid elements using Bentley's SmartSolid® with b-spline curves generated using Para Solids. Volumes and surface areas of concrete and steel are easily calculated.



**Program Name: LPILE Plus**

**Location:** Bridge Programs\Other\LPILE Plus

**Description:**

LPILE Plus (developed and marketed by Ensoft, Inc., Austin, TX) is a special purpose program based on rational procedures for analyzing a pile under lateral loading. Soil behavior is modeled with p-y curves internally generated by the computer program following published recommendations for various types of soils; alternatively, the user can manually introduce other p-y curves. Special procedures are programmed for developing p-y curves for layered soils and for rocks.

A single, user-friendly interface written for the Microsoft Windows© environment is provided for the preparation of input and analytical run and for the graphical observation of data contained in the output file. The program has been written in 32-bit programming codes for compatibility with the latest versions of the Microsoft Windows operating system. The program produces plain-text input and output files that may be observed and/or edited for their inclusion in project reports.

**Inputs:**

Several types of pile-head boundary conditions may be selected, and the properties of the pile can also vary as a function of depth.

**Outputs:**

The program computes deflection, shear, bending moment and soil response with respect to depth in nonlinear soils. Components of the stiffness matrix at the pile head may be computed internally by the program to help the users in superstructure analysis. Several pile lengths may be automatically checked by the program to help the user produce a design with an optimum pile penetration. LPILE Plus has capabilities to compute the ultimate-moment capacity of a pile's section and can provide design information for rebar arrangement. The user may optionally ask the program to generate and take into account nonlinear values of flexural stiffness ( $EI$ ) which are generated internally based on specified pile dimensions, material properties and cracked/uncracked concrete behavior.

**Program Name:** Mathcad®

**Location:** Bridge Programs\Other\Mathcad

**Description:**

Built on an intuitive whiteboard interface, Mathcad collapses two traditionally separate processes — design formulation and documentation — into one. Mathcad functions as an interactive worksheet that utilizes standard math notation and equation entry to solve problems, so no programming is required. Its patented whiteboard interface immediately returns or updates results, eliminating manual recalculation work.

**Inputs:**

NA

**Outputs:**

NA

**Program Name: MERLIN-DASH**

**Location:** Bridge Programs\Superstructure\Merlin Dash

**Description:**

MERLIN-DASH was developed for use by bridge design engineers who function in a production environment. MERLIN-DASH was developed to offer a wide range of features and options to meet the demands of universal usage in the analysis, design and rating of steel and reinforced concrete bridges.

The structural analysis is performed via a series of modular subroutines based on the stiffness method.

**Inputs:**

A mesh generation capability allows for the incorporation of fully automated AASHTO Dead Load (DL) and Live Load (LL) sequences.

**Outputs:**

MERLIN-DASH incorporates a flexible sequence of operations initiated with analysis and proceeding, at the user's option, to perform any or all combinations of the following functions for the AASHTO WSD, LFD or LRFD methods:

- Analysis — A complete analysis for all AASHTO DL and LL conditions with recycling for changes in sections due to design.
- Design — Determination of the size of steel structural components based on a user-controlled design sequence leading to either minimum cost or weight.
- Code Check — Complete and detailed code check of all steel or reinforced concrete beam components, which reference specific AASHTO equation numbers and applicable coefficients.
- Rating — Detailed inventory and operating rating of all beam components using either the AASHTO live load provisions or special user specified vehicles.
- Staging — Dead load pouring sequence stage analysis.

**Program Name: Opis****Location:** Desktop Icon**Description:**

Opis is AASHTOWare's next generation of bridge design software. BRASS-LRFD provides the system's structural analysis and specification checking engine.

**Inputs:**

Opis employs the same database and graphical user interface as Virtis.

**Outputs:**

Opis will provide a set of output reports to help the designer understand the performance of a new bridge. A tree-structured graphical representation of the LRFD Specifications indicate whether each article is passed or violated, and it provides access to the detailed calculations for the bridge and the specification text. A suite of X-Y plots shows moments, shears, deflections, actual vs. capacity envelopes, influence lines and other information. These will be incorporated into the report-writing feature currently under design.

**Program Name: Programmers File Editor (PFE)****Location:** Bridge Programs\Other\Programmers File Editor**Description:**

PFE is a large-capacity, multi-file editor. Although it is primarily oriented towards program developers and contains features like the ability to run compilers and development applications, PFE also makes a very good general-purpose editor for any function. It has all of the standard editorial features — insert or overwrite mode; open files; insert one file into another; etc.

**Inputs:**

NA

**Outputs:**

NA

**Program Name: Pontis****Location:** MDT Intranet**Description:**

Pontis is a comprehensive bridge management system developed as a tool to assist in the task of bridge management. Pontis stores bridge inventory and inspection data; formulates network-wide preservation and improvement policies for use in evaluating the needs of each bridge in a network; and makes recommendations for what projects to include in an agency's capital plan for deriving the maximum benefit from limited funds.

**Inputs:**

Pontis has been developed to provide the user with a well-organized and intuitive graphical user interface. The system consists of a set of modules, each of which has been designed to provide the user with the informational display, options and actions relevant to the module's particular function.

Pontis supports the entire bridge management cycle, allowing user input at every stage of the process. The system stores bridge inventories and records inspection data.

**Outputs:**

Once inspection data has been entered, Pontis can be used for maintenance tracking and Federal reporting. Pontis integrates the objectives of public safety and risk reduction, user convenience, and preservation of investment to produce budgetary, maintenance and program policies. Additionally, it provides a systematic procedure for the allocation of resources to the preservation and improvement of the bridges in a network. Pontis accomplishes this by considering both the costs and benefits of maintenance policies versus investments in improvements or replacements.

**Program Name: SEISAB****Location:** Bridge Programs\Other\Seisab**Description:**

SEISAB was specifically developed for the seismic analysis of bridges. The overall objectives in developing SEISAB were to provide the practicing bridge engineer with a usable design tool and vehicle for implementing the latest seismic design methodologies into the bridge engineering profession.

**Inputs:**

Horizontal alignments composed of a combination of tangent and curved segments are described using alignment data taken directly from roadway plans. SEISAB has generating capabilities that will, with a minimum amount of input data, automatically provide a model consistent with the model currently being used to conduct dynamic analyses. Seismic loadings in the form of response spectra are stored in the system and may be easily referenced by the user. The central theme underlying the development of SEISAB was to provide the bridge designer with an effective means of user-program communication using a problem-oriented language developed specifically for the bridge engineer. User input data is thoroughly checked for syntax and consistency prior to conducting an analysis and numerous default values are assumed for the data not entered by the user.

**Outputs:**

SEISAB contains both the single mode and multi-mode response spectrum techniques included in the AASHTO **Standard Specifications for Highway Bridges** and in the **Seismic Design Guidelines for Highway Bridges**. SEISAB can be used to analyze simply supported or continuous deck, girder-type bridges with no practical limitation on the number of spans or the number of columns at a bent. In addition, earthquake restrainer units may be placed between adjacent structural segments.

**Program Name: SIMON Systems**

**Location:** Available on disk

**Description:**

SIMON Systems is a PC software system for the design of straight steel plate-girder (I- or box-girder) bridges. The designs are according to the 16<sup>th</sup> Edition of the **Standard Specifications for Highway Bridges**. LRFD Specifications will be available in the future.

**Inputs:**

Program input can be made in either standard US or SI Units. For users who prefer a freeform screen-oriented input style to create SIMON input files, program SIP (Simon Input Processor) is also provided with SIMON Systems.

**Outputs:**

SIMON, the basic building block of SIMON Systems, is a line-girder program that will design up to 12 continuous spans that may contain hinges. The pre- and post-processing program SIMPLE is also provided as part of SIMON Systems. Program SIMPLE helps the user optimize the vertical web depth of an I- or box-girder.



**Program Name: System VANCK**

**Location:** Available on disk

**Description:**

System VANCK performs a V-Load analysis of a curved open-framed I-girder bridge system. VANCK then checks designated target girders in the cross section according to the provisions of the latest AASHTO **Guide Specifications for Horizontally Curved Highway Bridges**. The system is designed as a tool to simplify the preliminary design of curved steel I-girder bridges.

**Inputs:**

System VANCK utilizes input baseline stationing and cross-section information. It is limited to a minimum of two and a maximum of ten spans. In addition, the number of girders in the cross section must be at least two and not more than eight. Tangent segments and multiple radii of curvature are allowed but reverse curvature is not allowed.

**Outputs:**

In a single execution, System VANCK will determine the individual girder span lengths from input baseline stationing and cross-section information (program JINK), do a V-Load analysis of the curved-bridge systems (program VLOAD), and check the normal bending and tip stresses in the flanges and output fatigue stress ranges including the effects of lateral flange bending (program CURVCHK).

**Program Name: Virtis****Location:** Desktop Icon**Description:**

Virtis is AASHTOWare's new product for bridge load rating, featuring state-of-the-art graphical tools to speed preparation of the data and application of the results. Using BRASS as its proven analytical engine for load factor rating, Virtis provides an integrated database where rating inputs and outputs can readily be stored, reviewed and re-used. Through this database and the application-independent GUI, a user may provide a 3-dimensional description of a bridge superstructure. This bridge data, then, can be used by a variety of line-girder, 2-D or 3-D analysis packages, permit/routing systems and other third-party produced applications.

Although Virtis is written in C++, its support of the industry-standard COM interface makes it possible to access the system's data and functionality from many commercial software packages, including Visual Basic®, Excel®, AutoCAD® and Microsoft Word®.

**Inputs:**

As the successor to BARS, Virtis can import existing BARS files.

Data can be provided in either cross-section or schedule-based forms for steel girder bridges and cross-section based only for reinforced concrete bridges and schedule based for prestressed concrete bridges at this time.

Virtis contains a host of features to make load rating as easy as possible. Libraries of standard vehicles, loads, steel and prestressed shapes, load and resistance factors, materials, parapets and other bridge components allow bridge models to be built quickly in a drag-and-drop manner. Using the Windows clipboard, all or part of a bridge can quickly be copied to another bridge. As a bridge model is constructed, a graphical schematic framing plan, elevation view, cross-section view and other schematics provide feedback and make common types of errors apparent.

**Outputs:**

Virtis provides flexure and shear ratings, computes dead loads and distribution factors if they are not manually input, and analyzes deteriorated sections.