

AASHTOWare BrR Input - Prestressed Concrete Girder Bridge

This is an example of input for a typical NMDOT prestressed concrete (PSC) girder bridge. The purpose is to demonstrate AASHTOWare BrR input procedures & also give typical values used by the NMDOT. The procedure makes use of various “wizards” in the program to expedite model building.

The following dead loads are assumed for this example:

- Self-weights – internally generated by the program
- Railing & wearing surface – defined under “Typical Section”
- Slab thickening – may be either included in the deck thickness under “Typical Section”, or calculated externally and defined under “Member Loads”
- Haunch – input under the “Haunch Profile”, as a member load, or as a combination of the two.
- SIP forms – defined under “Member Loads”.

The available wizards include the Superstructure Definition Wizard, Diaphragm Wizard, & Superstructure Wizard (under Bridge Alternatives). In particular, the Superstructure Definition Wizard greatly simplifies input & reduces effort.

Continuous-for-Live Load versus Multi-span Simple spans: In a typical continuous-for-live load bridge, only one superstructure, comprising all continuous spans, is required. However, for multi-span simple span bridges, a unique superstructure must be defined for each span. In this latter case, the individual spans are linked together into a single superstructure (while maintaining the simple-span behavior) at the “Bridge Alternatives” definition screens, located at the end of the model input. Instructions are given for both bridge types.

AASHTOWare BrR has additional capabilities & input requirement not shown here & this example is not all inclusive.




PART 1 – INITIAL INPUT (DESCRIPTION, MATERIALS, GIRDER & APPURTENANCE DEFINITIONS)

Description

- **Bridge ID:** *Enter five-digit Br. No. & rater’s initials in the format of “00002_gwk”*
- **NBI Structure ID:** *Enter same value*
- **Bridge Completely Defined:** *Check prior to submitting finished model*
- **Description** *Include the following:*
 - *Short description of bridge type, number of spans, and span lengths*
 - *Name of raters*
 - *Date*
 - *Inspection report date*
 - *Project number(s) (from bridge plans)*
 - *Any unusual features*

Do not fill in other fields. They are populated when the bridge is imported into NMDOT’s database.

Materials

-  **Concrete**
*Add appropriate type(s). Use **Copy from Library...** button.*
-  **Reinforcing Steel**
*Add appropriate type(s). Use **Copy from Library...** button.*
-  **Prestress Strand**
*Add appropriate type(s). Use **Copy from Library...** button.*

- 📁 **Beam Shapes**
 - 📁 **Prestress Beam Shapes**
 - 📁 **I Beams**

Add appropriate type(s). Use **Copy from Library...** button.
- 📁 **Appurtenances**
 - 📁 **Parapet**

Add appropriate type(s). Use **Copy from Library...** button.
 - 📁 **Median**

Add as appropriate
 - 📁 **Railing**

Add appropriate type(s). Use **Copy from Library...** button.
 - 📁 **Generic**

Add as appropriate

PART 2 – SUPERSTRUCTURE DEFINITION WIZARD

- 📁 **SUPERSTRUCTURE DEFINITIONS**

Double-click the **Superstructure Definitions** item. At the **New Superstructure Definition** box click on the **Superstructure Definition Wizard** button.

Enter the appropriate data on the following screens:

Superstructure Definition Wizard – Superstructure Definition

Superstructure definition name: *PSC Girder Superstructure* (typical)
Superstructure Definition Type *Girder System*
Material type: *Prestressed (Pretensioned) Concrete*
Girder type: *PS Precast I*

Enter appropriate data for:

Number of spans:
Skew:
Number of girders:
Girder spacing =
Left overhang =
Right overhang =

LRFD analysis module: *AASHTO LRFD*
LFD analysis module: *AASHTO LFD*
Girder System Member Generation *Link Members*

Note: The “Link Members” item will create a unique definition for each of the 1st (exterior) & 2nd (interior) girders. All others will be linked (deactivated) to these two members.

Click **Next >** button

Superstructure Definition Wizard – Beam Data

Enter appropriate data for each span:

Span Length

Beam Shape

Strand Configuration *Harped* (typical)

Beam Projection, Left End & Right End *6"* (typical)

Click **Next >** button

Superstructure Definition Wizard – Continuous Supports

This screen appears only if the number of spans > 1 (multi-span continuous bridge). Enter pier support distances for each pier:

Support Distance on Left, SL *10"* (typical)

Support Distance on right, SL *10"* (typical)

Click **Next >** button

Superstructure Definition Wizard – Prestressing Strand

Materials, Strand Material: Select appropriate strand definition

Materials, Concrete Material: Select appropriate girder concrete definition

LRFD Stress Limits

The appropriate LRFD stress limits will appear after a concrete material is selected. Enter a value for the slab compression stress limit:

Final allowable slab compression: *2.4* (typical for 4000 psi deck concrete)

Click **Next >** button

Superstructure Definition Wizard – Deck

Deck Properties:

Enter appropriate deck data:

Actual deck thickness =

Structural deck thickness =

Effective flange width (LRFD) =

Deck concrete: Select appropriate deck concrete definition

Deck reinforcement: Select appropriate reinforcement definition

Note: The "Actual deck thickness" is used only for calculating deck weight. Slab thickening (older PSC girder bridges) can be added to the nominal deck thickness here.

Note: Many older bridge designs used a 1/4" deduction in the deck in the deck structural thickness in anticipation of future wear. Do not deduct this value in the "Structural deck thickness" above.

Haunch Type:

Enter appropriate data:

Haunch Dim. Y1 = *1* (typical)

Embedded height Y2 = *0* (typical)

Note: Do not use a haunch thickness > 1". The program uses the haunch data to increase section capacity. Because of the variable nature of actual haunch thicknesses, the NMDOT

limits the value used here to 1". For haunches > 1" (wide-flange beams), use the additional haunch dimension to calculate a member load & input under "Member Loads".

Click **Next >** button

Superstructure Definition Wizard – Loads

LRFD Live Load Distribution Factors *Leave these fields blank*

Uniform Loads

For "Uniform Loads", first click **New** & enter the following:

Load Case Name *SIP*

Type *D, DC*

Stage *Non-composite (Stage 1)*

Span *ALL Spans*

Load *Enter appropriate value*

Next, enter appropriate value for DC1 load due to slab thickening/haunch, if applicable.

Click **Next >** button

Superstructure Definition Wizard – Deck Template

Select the deck template:

Select the first template with only left & right outside barriers.

Appurtenances

For "Left Exterior" Appurtenance, enter the following values:

Name: *Select previously defined (concrete) parapet or (metal) railing*

Load Case: *DC2*

Measure To: *Front*

Edge of Deck Dist...: *Left Edge*

Distance At Start: *1.50*

Distance At End: *1.50*

Front Face Orientation: *Right*

Repeat above input for "Right Exterior" Appurtenance.

Sidewalks *Leave these fields blank*

Click **Finish** button to complete the Superstructure Definition Wizard.

PART 3 – VERIFY WIZARD INPUTS & COMPLETE SUPERSTRUCTURE INPUTS

Enter the following screens/items & verify data/input additional data as described

"PSC Girder Superstructure"

Definition *Verify data.*

Average humidity: *50%* (typical for NMDOT load rating)



**Framing Plan Detail
Layout** *Verify data*

*Right click on **Framing Plan Detail** & select **Schematic** to view graphical image of framing plan.*

Diaphragms

Run Diaphragm Wizard. Select diaphragm orientation from four available choices. Enter appropriate values in Diaphragm Wizard. Return to main diaphragm screen. Remove end diaphragms (not necessary) and add/modify values as appropriate.

Structure Typical Section

Deck *Verify data*

Deck (Cont'd) *Verify data & add the following data:*

Deck crack control...: 170

Sustained modular ratio factor: 2.00

Deck exposure factor: 1.00

Parapet *Verify data if applicable*

Railing *Verify data if applicable*

Lane Position *Verify data*

Wearing Surface

*Click **Copy from Library...** button & select appropriate wearing surface.*

Wearing surface thickness: *Enter appropriate value*

Load case: DW

Stress Limits *Verify data*

Prestress Properties *Verify data. Note: The “Loss method:” shown should be “AASHTO Approximate”. If not, revise. Also, add the following NMDOT standard time values:*

Transfer time: 15.0

Age at deck placement: 270

Final age: 3650

Shear Reinforcement Definitions

Vertical

Make a separate definition for each distinct stirrup configuration.

Name: Descriptive name

Material: *Select previously defined reinforcing steel definition*

Bar size: *Select appropriate value*

Number of legs: 2 (typical)

Inclination: 90.0 (typical)

MEMBERS

The “Superstructure Definition Wizard will have created a definition for each girder line under “Members”. Only “Member 1” & “Member 2” are active (can receive input), while the remaining members are inactivated by being linked to either of the active members.

Member 1

Member Loads

Uniform *Verify data*

Supports *Revise support constraints as required to conform with the support fixities shown on the Bridge Profile (Elevation) sheet in the Bridge Plans.*

*Right click on **Structure Typical Section** & select **Schematic** to view graphical image of typical section*

MEMBER ALTERNATIVES

Wizard Alternative

Member Alternative: *Exterior Girder* (change Member Alternative name for clarity)

Crack control parameter (Z): *170* (Enter for both “Top of beam:” & “Bottom of beam:”)

Exposure factor: *1.0* (Enter for both “Top of beam:” & “Bottom of beam:”)

Default rating method: *LRFR*

Default Units: *Verify data*

“Exterior Girder”

Default Materials: *Verify data*

Live Load Distribution

Click on “Compute from Typical Section” button.

Beam Details

Span Detail *Verify data*

Continuous Support Detail *Verify data if applicable (multi-spans only)*

Stress Limit Ranges *Verify data*

Slab Interface

Interface type: *Intentionally Roughened*

Default interface... *Check on*

Cohesion factor: *0.100 0.280*

Continuity Diaphragm

Ignore positive moment at supports in ratings *Check on*

Strand Layout

For each beam, click on the beam name to access the “Strand Layout” screen and verify/enter the following:

Description Type: *Strands in rows*

Strand Configuration Type: *Harped*

Symmetry: *Click on*

Click on “Mid span” button and define strand pattern.

*Next, click **Left end:***

Harp Point Locations Enter distance from girder end to harp point.

Leave “Radius” blank. On the girder cross-section graphic, select only the strand locations that represent the final left-end positions of the harped strands.

Repeat the above procedure for each girder.

Deck Profile

Deck Concrete

Click on “Compute from Typical Section” button. At “Computer Deck Profile...” screen, enter total deck thickness (no deduction for wearing surface)

Reinforcement (Applicable only for multi-span bridges)

Enter a set for each unique longitudinal bar along entire bridge length (there may be duplicate sets that use the same bar mark, depending on the actual reinforcement configuration). For each set, enter appropriate values for each of the following:

*Click **New** button*

Material: *Select appropriate Material as defined earlier*

Support Number: *[Redacted]*

Start Distance: *[Redacted]*

The input of strand locations is simplified by dragging the mouse across desired strand locations on the girder cross-section graphic, right-clicking & selecting “Add a strand at the selected positions...”

Continuous bar runs of smaller (#4 or #5) bars may be simplified by replacing with one long fictitious bar of the same size

For larger negative moment bars, ensure that start/end points & lap splices are accurately located in the model.

Length: [Redacted]
Std Bar Count: [Redacted]
LRFD Bar Count: [Redacted]
Bar Size: Select appropriate value
Distance: Enter distance from steel centroid to location selected under "Row"
Row: Select "Top..." or "Bottom..."
Bar Spacing: Enter spacing to nearest adjacent bar (not necessarily the bar mark spacing). Note: This value is only used for lap splice/development length calculations.

Haunch Profile *Verify data*

Shear Reinforcement Ranges

Vertical

The Stirrup Wizard simplifies data input. However, the wizard completely removes existing stirrup data. Therefore, it is important that all stirrup data for all spans is entered into the wizard prior to exiting the wizard. One exception is for identical spans (end spans for example) – these can be handled easily with the "Copy to Span..." button after completing the wizard.

Click on the "Stirrup Wizard..." button.

Span: 1

Start Distance: Enter distance from girder end to 1st stirrup

Note: Do not include the single leg #7/#8 bar at beam end. This bar is used to resist tension stresses.

For each set of stirrups/spaces, enter "Reinf. Name", "Number of Spaces", & "Spacing". Note: The "Finish by symmetry" method is not recommended in most cases.

Within the wizard, repeat the above steps for each unique span. When finished, click "Apply"

Examine stirrup data for all spans, modify as required, and copy data from any span to identical spans using the "Copy to Span..." button.

Click on "Exterior Girder", right click and select "Schematic" to view graphical image of stirrup spacings

Member 2

There are two possible ways to complete the input for "Member 2". One way is to repeat the above steps that were performed for "Member 1". A second, easier way (if both girders are identical) is to Copy & Paste the above "Member 1 Exterior Girder" definition to Member 2. This second method is shown herein.

1. Copy & paste "Member 1, Exterior Girder" to "Member 2": Highlight "Exterior Girder" under Member 1, right-click & select **Copy**. Highlight "MEMBER ALTERNATIVES" under "Member 2", right-click & select **Paste**.
2. Rename & make copied member alternative active: Double-click the pasted member alternative under "Member 2" & rename as "Interior Girder". Click **OK** to return to the main screen. Double-click "Member 2". Under the "Existing" & "Current" columns, click on each checkbox for "Interior Girder". This will simultaneously uncheck the boxes for "Wizard Alternative". Click **OK**.

3. Delete the Member 2, Member Alternative named “Wizard Alternate: Highlight “Wizard Alternate” under “Member 2”, right-click & select **Delete**.

4. Recalculate Live Load Distribution Factors: Under the “Interior Girder” member alternative,

Live Load Distribution

Click on “Compute from Typical Section” button.

5. Recalculate Effective flange width for “Interior Girder”:

Deck Profile

Deck Concrete

Click on “Compute from Typical Section” button. At “Computer Deck Profile...” screen, enter total deck thickness (no deduction for wearing surface)

PART 4 – COMPLETE MODEL

A. Copy Superstructure Definition (applicable mainly for multiple simple spans)

For a bridge that consists of multiple simple spans, each unique span must be defined as an individual “Superstructure Definition” using the preceding procedures.

In many cases, the variations between span data is similar enough that the copy command can be used to create additional spans with much less data input:

Move the pointer to the Superstructure Definition, “PSC Girder Superstructure” in this case. Right click and select “Copy”. Next move the pointer to “SUPERSTRUCTURE DEFINITIONS”, right click and select “Paste”. Highlight the newly created superstructure definition and rename, “Span 2”, for example. Review all input data for the new superstructure and modify as required. Note: After all manual revisions are made to the second superstructure definition, all “Compute...” buttons should be rerun to update values that may have changed.

B. Create BRIDGE ALTERNATIVE

For both single-span bridges & multi-span continuous-for-live load bridges, define one bridge superstructure as follows:

 **BRIDGE ALTERNATIVES**

Alternative Name: *Bridge Alternative* (typical)

Click **Superstructure Wizard...** button.

Number of superstructures Enter number of defined “Superstructure Definitions”

Prefix to Use When Generating Names

Click **Generate Superstructure Names** button

Click **Generate Superstructure Alternatives Names** button

In table below there will be a row for each defined superstructure defined previously under “Superstructure Definitions”. Under the **Superstructure Definition** heading, identify each defined Superstructure accordingly.

Click Finish & OK to complete the Bridge Alternatives process