

CT-Steel User Manual

CT-Steel Beta Version 3.0

Manual is under development, email Christopher.Patria@ct.gov, for any questions and issues.

1. Introduction

CT-Steel is an excel macro based program used to perform bridge load rating specification checking, using LRFR, for Steel I-Section flexural members. Force Effects are not computed by CT-Steel and are a user input into the program.

CT-Steel primarily consists of the Engine workbook, and the Member Input Book workbook. The Engine contains the macros which will organize the collection of Member Input Books which make up a bridge and contains the macros which will perform the ratings. The Engine opens the Member Input Books, reads the Member Data stored within the Member Input Books, computes the rating calculations and outputs SpecCheck xml files and the schematic file. The Member Input Book can be thought of a model.

1.1. Initial Setup

CT-Steel is delivered as Zip archive. CT-Steel can retain a database of Bridges, and therefore it is ideal to have one active copy of CT-Steel in one location; the Member Input Books should be stored with the bridge's calculations files. Depending on your organizations file storage structure, CT-Steel should be stored on a shared drive. The drive can be network drive or a cloud drive, e.g. SharePoint or OneDrive. Having a centralized location will help collaboration between engineers within your organization. Note, special instruction for cloud drives can be found in the Datasources section. You may also elect to store CT-Steel locally. CT-Steel can be opened and operated as read-only.

1.1.1. Step By Step Instructions

1. Extract the contents of the CT-Steel.zip to the intend permanent storage location
 2. Open CT-Steel.xlsm
 3. Click Enable Contents to enable macros.
- Note the user may need to right click the excel file, click properties, and unblock file.

1.2. Update Instructions

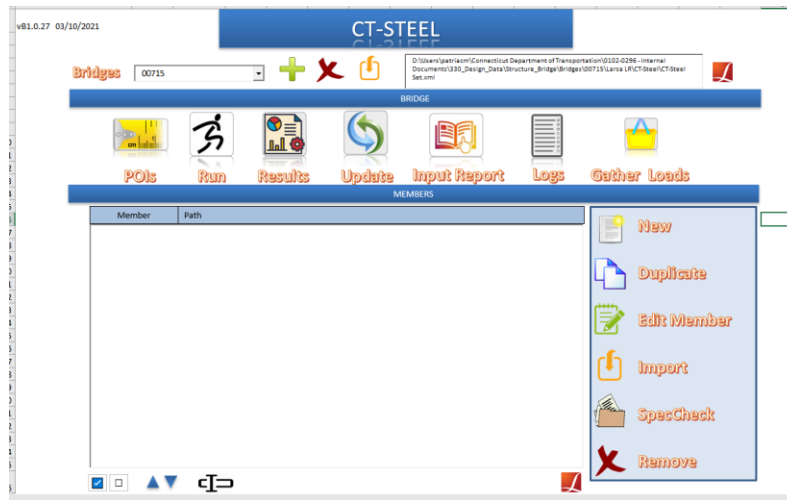
CT-Steel is designed such that the program can be updated without having to re-input member data. Copy the contents of the new version of CT-Steel.zip and paste within the same folder where the CT-Steel engine files currently reside.

1.2.1. Step By Step Instruction

1. Open the CT-Steel.zip archive.
2. Copy all files in CT-Steel.zip archive
3. Paste and override files in older version of CT-Steel in your file system

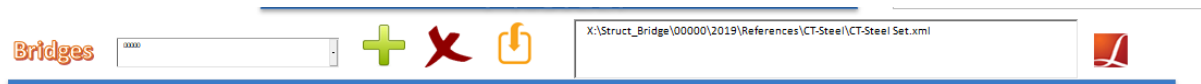
Note, a cheat sheet is included in the CT-Steel.zip folder called *Update Instructions.pdf*

2. Control



The Control is broken up into three groups of commands, Bridge Commands, Engine Commands, and Member Commands.

2.1. Bridge Commands



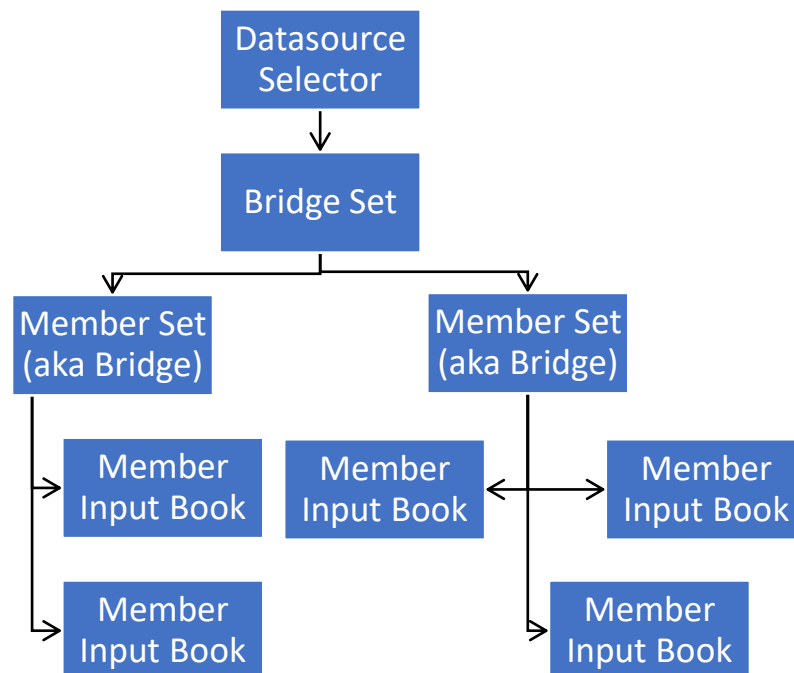
The bridge commands organize a collection of Member Input Book. A collection of Member Input Books will make up a bridge.

2.1.1. Datasource Selector

To open, click the "Bridges" Icon:



The datasource selector is used to point to CT-Steel to the Bridge Set file. The bridge set file is created by CT-Steel automatically. The Bridge Set file contains a list of file paths to each bridge's Member Set file. The Member Set File contains a list of file path to each member input book for that bridge.



2.1.1.1. File Structure

2.1.1.1.1. Data-Sources

Stores the file path of the active database.

File Location: User's Windows Profile / STL-Datasources / STL-Datasource.xml

2.1.1.1.2. Bridge Set

The bridge set is the database which contains files path to each member set file. The Member Set can be thought of a bridge, i.e., a collection of members.

File Location: [Pointer from STL-Datasource] / STL-Sets / BridgeSets.xml

2.1.1.1.3. Member Set

The member set is a database which contains file paths to each member input book for the bridge.

2.1.2. Bridge Selector



This drop down will set the active Bridge from the Bridge Sets.

2.1.3. Create New Bridge



This button will create a new bridge record in the bridge set database and set it active.

2.1.4. Remove Bridge



The button will remove the bridge from the database. Bridge Files will not be deleted, just the pointer to the files will be removed from the database.

2.1.5. Import Bridge to Database

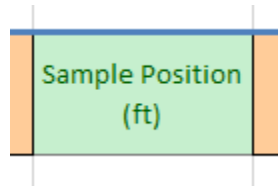


This button will allow the user to import an existing bridge to the current database. Note: the bridge files will not be duplicated, only a pointer to those files will be added to the database.

3. Member Input

3.1. Cell Types

3.1.1. Variable Parameter Cells

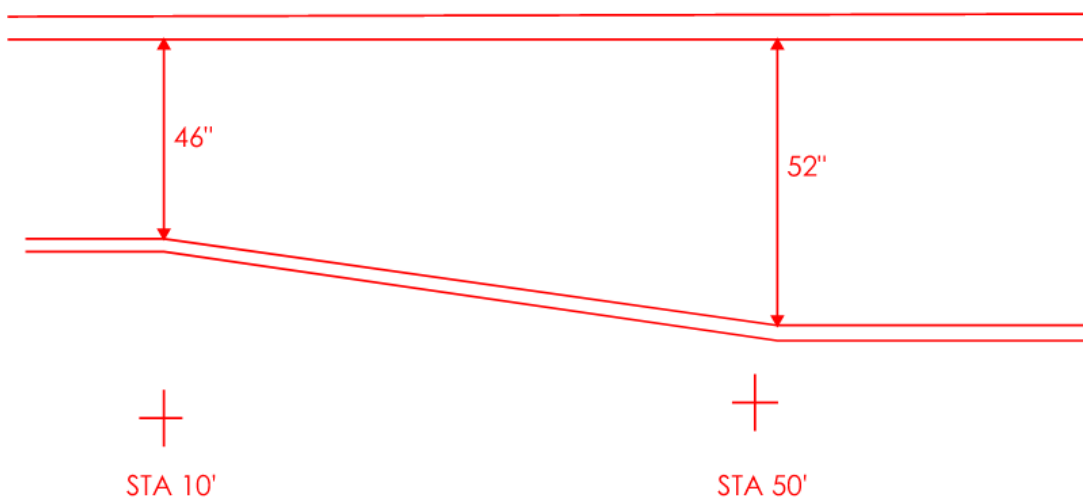


These cells are identified by a heading with a green background. During runtime CT-Steel will enter the station or web depth into the cell and read the remaining values in that row, except for the start and end distance. The user can write an excel function which reference the variable parameter cell to defining a varying input, such as web depth variation as a function of the station. The Sample Web Depth input is commonly used to change the datum where a feature is define along the web depth.

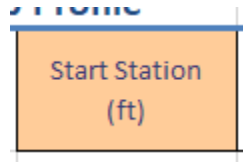
Example:

The below example demonstrate a web depth variation from station 10' to station 50' for a web increasing in depth 6" over 40'.

G3 : $=46+6*(F3-B3)/C3$						
1	Web Profile					
2	Start Station (ft)	Length (ft)	End Station (ft)	Thickness (in)	Sample Position (ft)	Height (in)
3	10	40	50		50	52

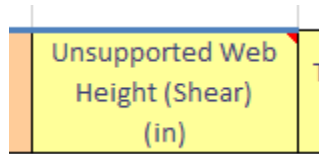


3.1.2. Required Cells



These cells are identified by a heading with an orange background. The user should enter values in these cells. For row based input, the user is not required to input rows, however, when a row added all columns with required cells should be populated.

3.1.3. Situational Dependent Required Cells



These cells are identified by a heading with a yellow background. The input into these cells are not required for all member types and cases.

3.2. Stages

The user should establish a construction staging sequence when certain components of the cross-section are considered effect and when certain load cases are applied to the member. There is no limit to the number of stages. The web and flanges are automatically assigned to be active in stage 1. The deck reinforcing is automatically assigned to the deck hardening stage. CT-Steel will automatically create a new stage after the end of all stages to apply live load.

The stage assigned to the components shall be matched by the user to the dead load case assignment.

As an example, a composite girder bridge is initially constructed. In a later rehibition project, cover plates are added to increase the moment capacity. Subsequently the wearing surface was increased in a resurfacing projected.

Stage	Component Stage Assignment	Loads Stage Assignment
1	Web (Auto), Flanges (Auto)	Structural Steel Weight, Deck Weight
2	Deck Hardened	Parapets Loads, 2.5" bit conc. Load, Cover Plate Loads
3	Cover Plates	
4		2" additional bit conc. load
5		Live Loads (Auto)

3.3. Bridge

Parameter	Input Options	Remarks
Bridge No.	Label	Enter the Bridge ID number.
Carries	Label	Enter the feature carried by the bridge.
Crosses	Label	Enter the feature carried by the bridge.
Span	Label	Enter the span number the member is located in per the bridge inspection report.
Member	Label	Enter the bridge log name from the bridge inspection report.
Curved	Boolean (TRUE or FALSE)	Indicate if the member is horizontally curved. Used to determine if the web is compact.
Member Type	Label	Added to the rating results outputs. Examples: Girder, Floorbeam, Stringer
Kinked	Boolean (TRUE or FALSE)	Indicate if the member is horizontally kinked or continuously chord. Used to determine if the web is compact.
isLongitudinal	Boolean (TRUE or FALSE)	Indicate if the member is longitudinal or transverse member. Used in fatigue calculations.
System Factor	Decimal	Enter the system factor for the member.
Condition	Decimal	Enter the condition factor the member.

Note:

1. The bridge no., span no., and member will be used as the folder name for the spec check. The combination of these inputs needs to be unique.
2. System Factors and condition factors can be overridden with the ControlOptionRange input.
3. The system factor will not be applied to shear ratings, except when explicitly overridden in the ControlOptionRange input.
4. Curved and Kinked members are not typically allowed to be evaluated using Appendix A6. CT-Steel does not automatically disallow consideration of Appendix A6 for these member types. The user should use the control options to disable consideration Appendix A6. Conversely, CT-Steel will consider these webs non-compact which will automatically disallow consideration of the plastic moment capacity for composite section in positive flexure.

3.3.1. Span



3.3.1.1. Span Information

Define the stationing along the member (center line of the web). This stationing is not of the roadway, but of the member itself. This stationing is a user defined stationing and should be used for the most intuitive entry (usually making the first bearing line station 0). Add rows to this table for each span. The spans should be the structural spans of the member.

Parameter	Input Option	Remarks
Structural Span Number	Integer	Enter an integer of the span.
Span Length	Decimal	Enter the length between points of support for the member.
Gap at right	Decimal	Enter the gap between supports. This provides a gap in the stationing for the rating. See example below.
Start Station	Decimal	Enter the station for the start of the span.
End Station	Decimal	Enter the station of the end of the span.
Parent Span	Integer	Enter the parent span. This is used for the output of the spec checks. For example, a floorbeam in the Bridge

Notes:

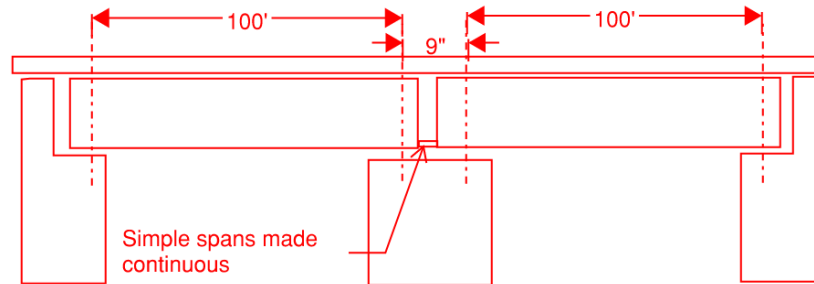
1. For curved members the stationing is measured along the arch length of the centerline of the web.

Examples:

The below example is of a three-span continuous girder with spans 118'-10, 221'-6", & 118'-10".

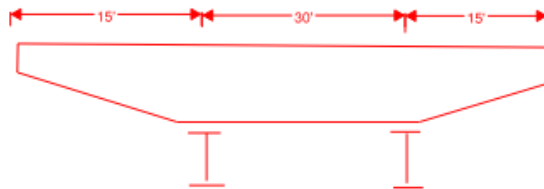
Span Information				
Span Number	Span Length (ft)	Gap at right (ft)	Start Station (ft)	End Station (ft)
1	118.8333333		0	118.8333333
2	221.5		118.8333333	330.3333333
3	118.8333333		330.3333333	449.1666667

The gap is used to define a gap in stationing between support lines. The engine will not rate the POIs within this range. See example below:



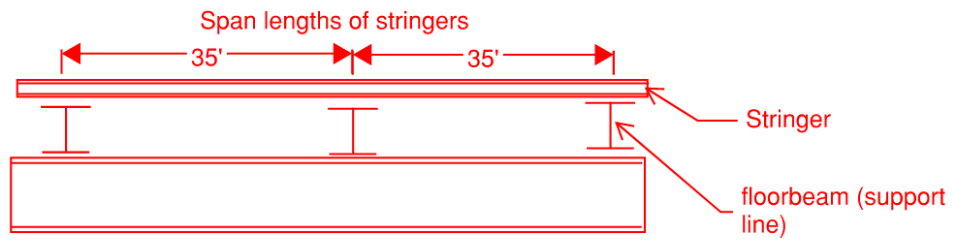
Span Information				
Span Number	Span Length (ft)	Gap at right (ft)	Start Station (ft)	End Station (ft)
1	100	0.75	0	100
2	100		100.75	200.75

Floorbeam example:



Span Information				
Span Number	Span Length (ft)	Gap at right (ft)	Start Station (ft)	End Station (ft)
1	15		0	15
2	30		15	45
3	15		45	60

Stringer example:

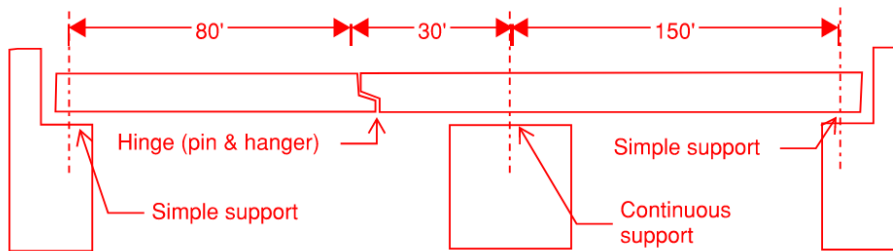


Span Information				
Span Number	Span Length (ft)	Gap at right (ft)	Start Station (ft)	End Station (ft)
1	35		0	35
2	35		35	70

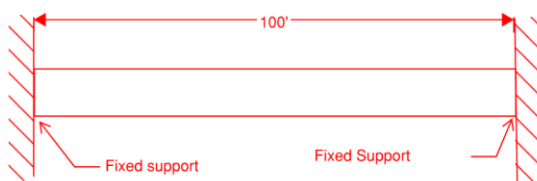
3.3.1.2. Hinges & Simple Supports



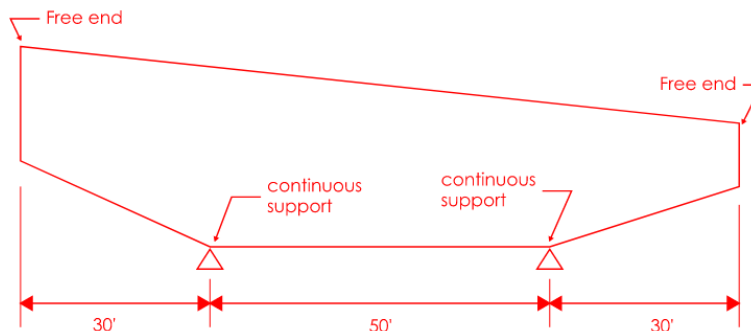
Enter the Station for each location of a simple support, hinge, continuous support, free end, fixed end. This data is used to determine continuity for Plastic Moment Capacity calculations and determine the End Panels for shear. A simple support or hinge is any point along the member where there is a moment release in the girder. See below examples.



Boundaries	
Station (ft)	Boundary Type
0	Simple
80	Hinge
110	Continuous
260	Simple



Boundaries	
Station (ft)	Boundary Type
0	Fixed
100	Fixed



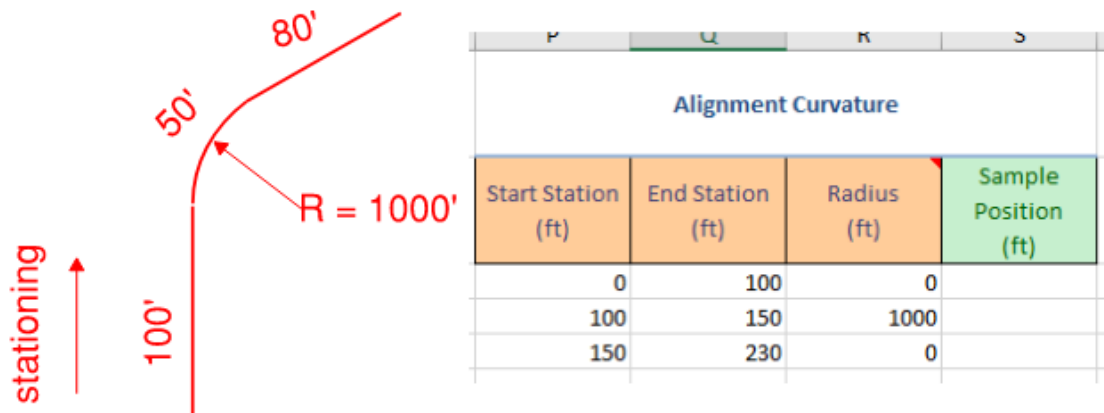
Boundaries	
Station (ft)	Boundary Type
0	Free
30	Continuous
80	Continuous
110	Free

For legacy models with only the locations of hinges or simple supports defined, CT-Steel, will assume any entered station under boundaries without a boundary type specified is simple support and the starts and ends of spans, without a station defined in the boundary conditions, is a continuous support. See User Manual for additional examples for entering the boundary condition data.

3.3.1.3. Alignment Curvature



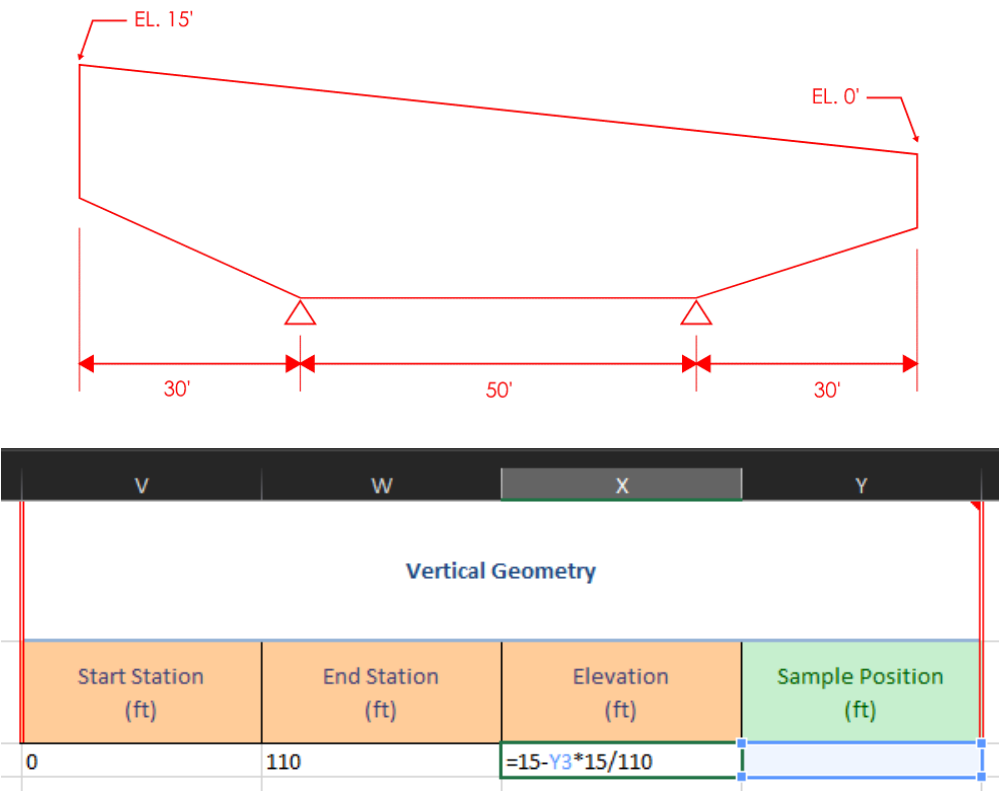
Enter the curvature of the alignment. Input is not required for tangent girders. The user can enter an equation into radius cell to define non-constant radius (spirals). See example below:



Note: In the above example, the user may omit the first and third row from the table.

3.3.1.4. Vertical Geometry

This entry specifies the vertical geometry of the girder web. CT-Steel will use this information to determine the chord angle of the varying flange when determining LTB parameters for non-prismatic girders. When left blank, CT-Steel will assume the top of the web is at a constant elevation and any variation in web depth is achieved by varying the elevation of the bottom of the web/bottom flange.



3.3.2. Traffic



Enter traffic data. This information is used in fatigue rating calculations only.

Parameter	Input Option	Remarks
$(ADTT_{\text{One Direction}})_{\text{PRESENT}}$	integer	Enter the present one directional ADTT.
Number of stripped Lanes	integer	Enter the number of stripped lanes in one direction
Growth rate, g	Percent	Enter the traffic growth rate.
Bridge Carries two directions	Boolean (TRUE or FALSE)	Indicate if the bridge carries bidirectional traffic.
Directional percentage	Percent	Enter the direction percentage. For bridges which carry bidirectional traffic enter the directional percentage for traffic in one direction. For bridge which carry traffic in a single direction, enter 100%.
$(ADTT)_{\text{PRESENT}}$	integer	Enter the present day total ADTT carried by the structure.
$(ADTT_{\text{SL}})_{\text{PRESENT}}$	integer	Enter the present-day single lane ADTT.
$(ADTT_{\text{SL}})_{\text{INITIAL}}$	integer	Enter the initial ADTT single lane when the bridge was constructed (or fatigue detail in service).
$(ADTT_{\text{SL}})_{\text{LIMIT}}$	integer	Enter the maximum single lane ADTT carried by the route.

Note: Bridge carries two directions, Directional Percentage, and $(ADTT)_{\text{PRESENT}}$ are inter related.

3.4. Control Options



Enter the control options in the *Value* column. Note these control options will apply to the entire member. Note that some of the control options can be overridden across specific ranges when defined in the *Control Option Ranges* section.

3.5. Control Option Ranges

Member Book

Bridge

ControlOptionRNGs

The user can specify certain ranges to modify the control options applied in the previous section.

Parameter	Input Options	Remarks
Stat Station	Station	Enter the start station for the range.
End Station	Station	Enter the end station for the range.
Control Option	VB Variable	Enter the name of the Control Option to be overridden for the range and rating event. The name of the control option should be exactly as it appears in the VB Variable Cell in the ControlOptions table. See the ControlOptions table for applicable control options, See Notes below as well.
Value	Value	Enter the override value for the control option. See the ControlOptions table for applicable values.
Force Effect	M M+ M- V N/A [blank]	Indicate which force effect the control option is to be overridden for. See Note 3 below. If left blank or set to N/A the control option override will apply to all limit states.
Limit State	STR SVC N/A [blank]	Indicate the limit state the control option applies to. If left blank or set to N/A [blank] the control option override will apply to all limit states.

Notes:

1. The right-hand columns of the ControlOptions sheets (previous section) indicates which control options can be overridden in the ControlOptionRanges input. The Force Effect Options column of the ControlOptions table indicates whether a certain control option can be overridden for specific force effects. Some control options, such as asComposite cannot be overridden independently for negative or positive moment, meaning that the asComposite control option range will apply both to negative and positive moment rating events.
2. In addition to the available control options as indicated on the Control Options sheet, "ConditionFactor" and "SystemFactor" can also be entered as a Control Option to be overridden.
3. If a system factor is overridden with the force effect cell equal to "V" or "N/A", CT-Steel will apply also apply to shear ratings, meaning, the system factor for shear may not be taken equal to one. Note if force effect is blank, CT-Steel will not apply the system factor entry to the shear rating.
4. Note that the Limit State entry is not shaded, and therefore not used/not required, as at this time, each control option is only applicable to a specific limit state.

Examples:

The below example will disable the plastic moment capacity and appendix A6 between station 50' & 100' for positive moment. This might be used for situations where there may be holes in the bottom flange in this location.

	A	B	C	D	E	F	G	H
1	Control Option Ranges							
2		Start Station (ft)	End Station (ft)	Control Option	Value	Force Effect	Limit State	
4		50	100	isPlastic	FALSE	M		
5		50	100	allowA6	FALSE	M		
6								
7								

3.6. Capacity Overrides



This table is used to add user calculated capacities to be considered in the rating. This can be used to add details that CT-Steel cannot rate, such as beam ends or field splices. This can also be used to override CT-Steel capacities.

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range. See Note 1 below.
Force: Shear/Moment	Shear Moment	Indicate if this capacity override is for shear or moment.
Min/Max/Abs	Min Max Abs	The extrema to define this rating event. Min: CT-Steel will apply the capacity override for negative live load force effects only. Max: CT-Steel will apply the capacity override for positive live load force effects only. Abs: CT-Steel will apply the capacity override for positive and negative live load force effects.
Limit State: Strength/Service	Strength Service	Enter the limit state to define this rating event.
Capacity	Value	Enter the user defined capacity for this defined rating event. For Shear capacity overrides, the unit is KIP, and for Moment capacity override, the unit is KIP-IN.
Override=True Append=False	Boolean (TRUE or FALSE)	Set to TRUE to replace CT-Steels ratings for the defined rating event. Set to FALSE to add additional failure modes for the defined rating event. See Note 2 below.
Sample Position		CT-Steel will enter the station in this cell. See Article 3.1.1 Variable Parameter Cells
Failure Mechanism	Text	Enter the name of the failure mechanism. This name will be added to the spec check output.

Example:

A	B	C	D	E	F	G	H	I	J	K	L
Capacity Override											
	Start Station	End Station	Force: Shear/Moment	Min/Max/Abs	Limit State: Strength/Service	Capacity: Kip/kip-in	Override=True Append=False	Sample Position (x)	Failure Mechanism		
	40.5 B		Moment	Abs	Strength	110638	FALSE		Flange Splice - DETAIL 21		
	40.5 B		Shear	Abs	Strength	1090	FALSE		Web Splice		
	81.125	100	Moment	Min	Strength	1090	TRUE		Override AASHTO LTB equations		
	140 B		Moment	Abs	Strength	98370	FALSE		Flange Splice - DETAIL 22		

Notes:

1. "L", "R", "B" can be used in the End Station to set the capacity override to a single POI. Where "L" denotes the left side of the station, "R" denotes the right side of the station, and "B" denotes both sides of the station.
2. Override/Append Setting

Append Mode: CT-Steel will compute capacities and perform ratings for the built-in rating articles and also compute ratings for the entered capacity override.

Override Mode: CT-Steel will not compute capacities for the rating event (Force Effect, Extrema, Limit State) and only use the entered capacity.

3.7. Materials

3.7.1. Steel

This table is used to define material definitions for steel components (structural steel and reinforcing steel).

Parameter	Input Options	Remarks
Name	Text	Enter the name of the steel material. This is the name that the user will enter when assigning a material to component.
Fy	Value	Enter the yield strength of the material
Fu	Value	Enter the ultimate strength of the material. Required only when holes are defined in the flange and cover plates inputs.
E	Value	Enter the modulus of elasticity of the material.

3.7.2. Concrete

This table is used to define material definitions for concrete slab.

Parameter	Input Options	Remarks
Name	Text	Enter the name of the concrete material. This is the name that the user will enter when assigning a material to component.
Fy	Value	Enter the compressive strength of the material.
Fr	Value	Not used by the engine.
E	Value	Enter the modulus of elasticity of the material.

3.8. Slab

3.8.1. Effective Flange

Input in this table is only required for composite members or for members with the asComposite control option enabled.

Parameter	Input Options	Remarks
Start	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End	Station	Enter the end station for the range.
Deck Thickness	Dimension	Enter the structural thickness of the deck slab.
Sample Position		CT-Steel will enter the station in this cell. See Article 3.1.1 Variable Parameter Cells
Width at Right	Dimension	Enter the effective flange width on the right side of the web. CT-Steel will add this input with the width at left input to determine the total effective flange width.
Width at Left	Dimension	Enter the effective flange width on the left side of the web. CT-Steel will add this input with the width at right input to determine the total effective flange width.
Material	Name	Enter the Name of the concrete material for this range. The name shall match exactly one of the name of the material entered on the Materials > Concrete sheet.

Notes: CT-Steel will sum the width at right and width at left inputs to determine the total effective flange width for composite section properties. Splitting the input for the left and right is to simplify user input.

3.8.2. Composite

Input in this table is only required for composite members or for members with the asComposite control option enabled.

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Composite or NonComposite	C NC	Enter C for composite or NC for non-composite. The user does not need to enter NC as CT-Steel will assume that any range without a definition for composite action is non-composite. NC is only required for ranges where unintended composite action is desired in combination with the asComposite control option set to TRUE.
Harden Stage	integer	Enter the Stage at which the deck slab will be considered effective.

3.8.3. Reinforcing

Use this table to enter ranges for deck reinforcement. Top and bottom mats are entered as separate rows. Ranges may overlap, see note below table.

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Area	Value	Enter the total area of steel for the mat for this range.
Depth from TOP of Deck	Distance	Distance from the top of the structural deck to the center of area for the reinforcing mat.
Material	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the name of the material entered on the Materials > Steel sheet.
Bar Row	T B	Enter T to indicate this entry row is for the top mat, enter B to indicate this entry row is for the bottom mat.

Note:

1. If an overlapping range is entered for a given bar row (top or bottom mat) CT-Steel will sum the areas for the overlapping cross sections and determine the resultant depth from TOP of Deck for that mat.
2. The top and bottom mat are handled as separate elements when determining section properties.
3. Development lengths should be accounted for in the start and end stations inputs.

3.8.4. Haunch

Use this table to specify haunch height ranges. If a range is not specified, CT-Steel will assume a haunch height of zero. The top of web is used as the datum. Ranges are not required for non-composite ranges, except for situation where the user enables the asComposite control option.

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Height	Value	Enter the distance from the top of the web to the bottom of the structural slab.
Sample Position		CT-Steel will enter the station in this cell. See Article 3.1.1 Variable Parameter Cells

3.9. Girder



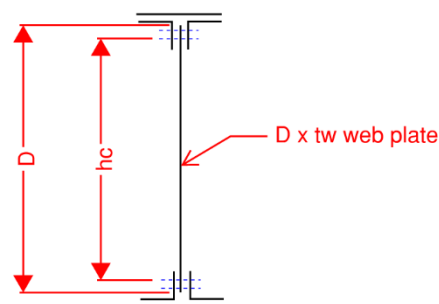
3.9.1. Web



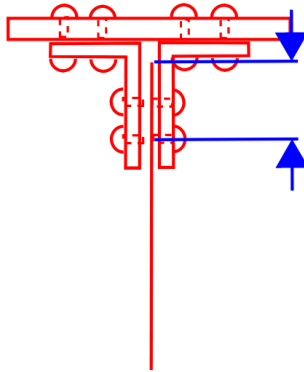
Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Thickness	Dimension	Enter the web thickness
Sample Position		CT-Steel will enter the station in this cell. See Article 3.1.1 Variable Parameter Cells
Height	Dimension	Enter the depth of the web plate. For rolled or welded section, this is the distance between flanges. For mechanically fastened built-up sections this is the total depth of the web plate.
Material	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the name of the material entered on the Materials > Steel sheet.
Unsupported Web Height (Shear)	Dimension	Optional input to override the web height used to compute the compute web slenderness in shear buckling equations. See Note 1.
Top Support Line	Dimension	Optional input to indicate the distance at the top of the web plate which is restricted from buckling. See Note 2.
Bottom Support Line	Dimension	Optional input to indicate the distance at the bottom of the web plate which is restricted from buckling. See Note 2.

Notes:

1. The unsupported web height, h_c shown below, is the distance between fastener lines in built-up members. CT-Steel will compute the shear buckling to shear yielding ratio, c , using h in the slenderness equations for shear, similar to the AISC approach.



2. The distance from the edge of web to the point where the web can be assumed to be supported from buckling used in flexural calculations to reduce web slenderness.



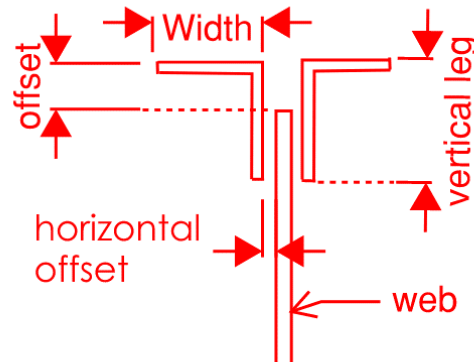
3.9.2. Flanges

Applicable for both top and bottom flanges. A flange must be defined throughout the entire length of the member. Overlapping flange ranges are not allowed.

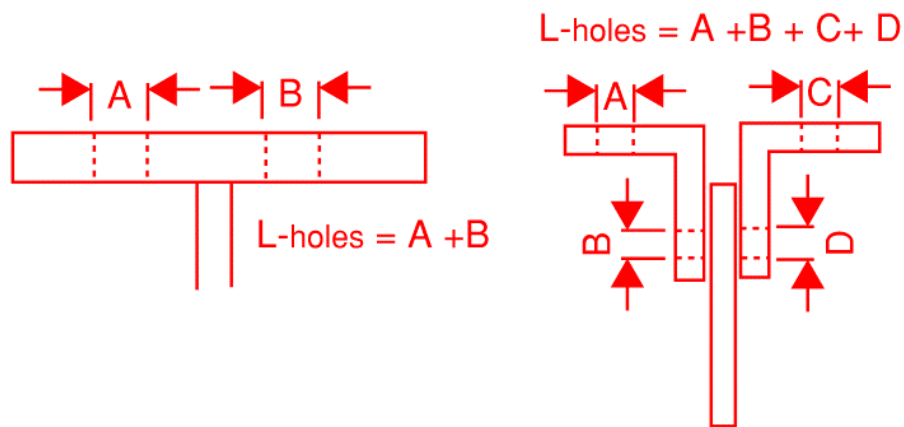
Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Thickness	Dimension	Enter the flange thickness
Sample Position	REF	CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Width	Dimension	Flange Width. For flanges of welded plate girders and rolled sections, enter the overall flange width. For mechanically fastened built up members with angle flanges, enter the horizontal leg width of one of the angles. See Note 1.
Material	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the name of the material entered on the Materials > Steel sheet.
Vertical Leg	Dimension	Leave cell blank for welded plate girder and rolled section flanges. Enter the vertical width of angle flanges for one of the angles. See Note 1. Note: Entering any value, including a 0 will trigger CT-Steel to assume this is angle flange and double the flanges.
Offset	Dimension	Leave cell blank for welded plate girder and rolled section flanges. Enter the distance from the back of the angle flange leg to the edge of the web plate.
L-holes	Dimension	If holes are present in the flange, enter the length of holes along the fracture plane. CT-Steel will use the entered thickness to compute the net area. The staggered hole reduction rule should be considered when determining this value. See Note below.
Horizontal Offset	Dimension	Leave cell blank for welded plate girders and rolled section flanges. Enter the horizontal distance from the edge of the web to the edge of the vertical leg of the angle flange. Typically equals zero when the vertical leg of the flange is in contact with the web. See Note 1 below.

Notes:

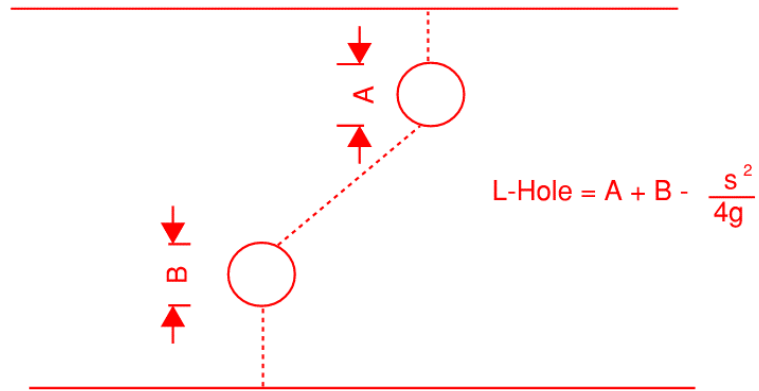
1. Input schematic for angle flanges:



2. Enter the sum of the diameters of holes along the net section for tension flange fracture. CT-Steel will multiple the *L-holes* by the thickness of the flange when computing the net area of the flange. This input is the entire input, meaning for riveted built-up members entire the sum of all diameters of both angles. Note: CT-Steel will not automatically disable allowA6 or allowPlastic if a value is provided. The user should manual disable these control options.



If the critical net section is along staggered hole path, the L-Hole value shall account for the additional width per BDS 6.8.3. See schematic below.



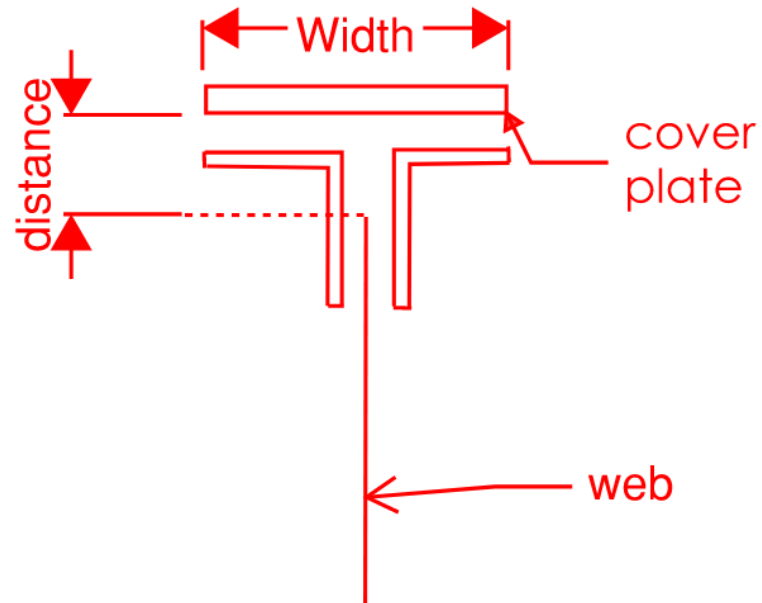
3.9.3. Cover Plates



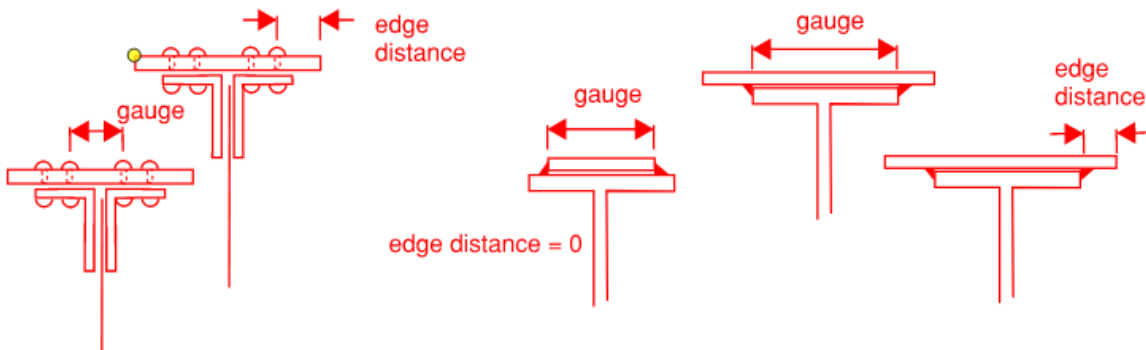
Parameter	Input Option	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Thickness	Dimension	Enter the flange thickness
Sample Position	REF	CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Width	Dimension	Enter the width of the cover plate
Material	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the names of the material entered on the Materials > Steel sheet.
Active Stage	Integer	Enter the stage at which the cover plate is effective.
Distance from TOP/BTM of CP to BTM/TOP Web	Dimension	Enter the distance from the interior edge of the cover plate to the edge of the web. See Note 1.
Gauge	Dimension	Enter the gauge width of fasteners or welds attaching the cover plate to the flange. See Note 2.
L-Holes	Dimension	If holes are present in the cover plate, enter the length of holes along the fracture plane. CT-Steel will use the entered thickness to compute the net area. The staggered hole reduction rule should be considered when determining this value. See Note in the flange input section above.
Start Development Length	Distance	Enter the distance from the start station that the cover plate is fully developed.
End Development Length	Distance	Enter the distance to the end station where the cover plate is fully developed at the end of the range.
Edge Distance	Dimension	Enter the width from the connection to the flange to the free edge of the cover plate. See Note 2.

Notes:

1. Cover plate input schematic:



2. The Gauge & Edge Distance inputs are used to determine the "b" value to compute the b/t ratios for cover plates in the local buckling equations.



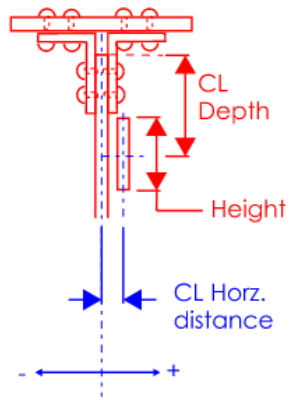
3.9.4. Side Plates

These tables are used to defined vertically oriented plates attached to the webs near either the top flange or bottom flange. Note if side plates are present on both sides of the web plate at a given flange, two entries are required for each of those plates. Overlapping ranges are acceptable.

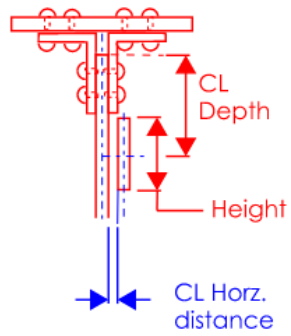
Parameter	Input Option	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Thickness	Dimension	Enter the side plate thickness
Height of Plate	Dimension	Enter the height of the side plate. See Note 1.
CL Depth From TOP/BTM of Web	Dimension	Enter the vertical distance from the center of the side plate to edge of the web. A positive value positions the side plate towards the center of the web a negative value positions the side plate away from the center of the web. See Note 1.
CL Horz. Distance from CL of Web	Dimension	Enter the distance from centerline of the web to centerline of the side plate. Positive values indicate the side plate is on the right face of the web, and negative values indicate the side plate is on the left face of the web. The expect input for this cell can change from the above description based on the entry provided in Alternate Method for Horizontal placement cell. See Note 1.
Material	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the names of the material entered on the Materials > Steel sheet.
Active Stage	Integer	Enter the stage at which the side plate is effective.
Start Development Length	Distance	Enter the distance from the start station that the cover plate is fully developed.
End Development Length	Distance	Enter the distance to the end station where the cover plate is fully developed at the end of the range.
Sample Position	REF	CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Alternate Method For Horizontal Placement	[Blank] 1 -1	[Blank] = The horizontal distance is measure as described above for CL Horz. Distance from CL of Web. 1 = the horizontal distance is measured from the inside face of side plate to outside face of the web plate. And the web plate is placed on the right side of the web. -1 = the horizontal distance is measured from the inside face of side plate to outside face of the web plate. And the web plate is placed on the left side of the web. See note 1.
Sample Web Depth	REF	CT-Steel will enter the web depth of each POI in this cell. See Article 3.1.1 Variable Parameter Cells

Notes:

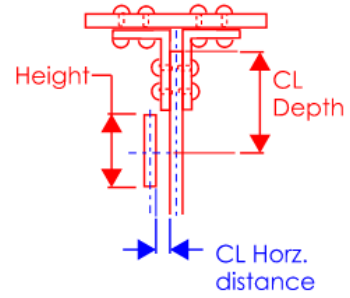
1. Schematic:



With Alternate
Method = [Blank]



With Alternate
Method = 1



With Alternate
Method = -1

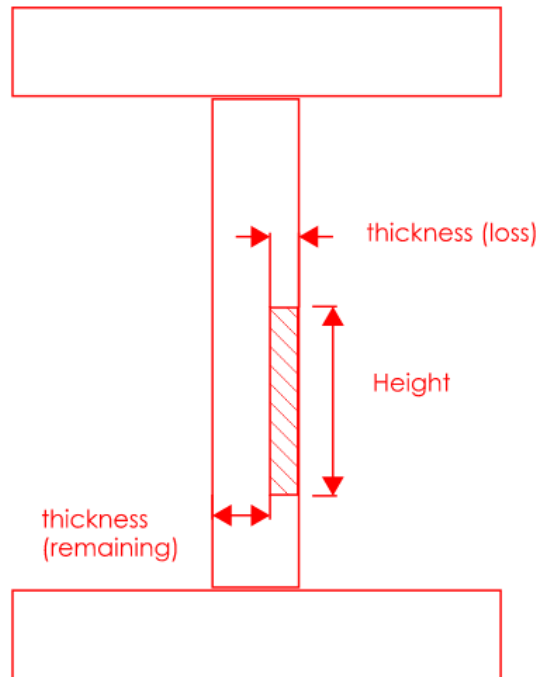
3.10. Section Loss

3.10.1. Web Section Loss

Parameter	Input Option	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Sample Position	REF	CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Thickness	Dimension	Loss Type = SL: Leave Blank Loss Type = L: Enter the thickness that was lost within the height inputted. Loss Type = R: Enter the remaining thickness within the height inputted.
Height	Dimension	Loss Type = SL: Leave Blank Loss Type = L or R: Enter the height the thickness loss or remaining input applies to.
Percent Thickness Loss	Percentage	Loss Type = SL: Enter the percent section loss for the total web height. Loss Type = L or R: Leave Blank
Loss Type	L R SL	L = Have CT-Steel compute the section loss percentage based on loss inputs. R = Have CT-Steel compute the section loss percentage based on remaining inputs. SL = User defined section loss percentages.

Notes:

1. CT-Steel compute an average web thickness based on the entered section losses to use in the rating calculations, i.e., smear the loss throughout the web area.
2. Schematic for Loss and Remaining Section Loss Type input:



3. Basic Section Loss equation to compute Section Loss Percentage for Loss Type = SL:

$$SL\% = \frac{Height * thickness(loss)}{Web Depth * Web Thickness} * 100\%$$

4. Overlapping ranges can be entered. CT-Steel will sequentially apply the section losses. Programmatically, the procedure is CT-Steel will start with the original dimension entered in the Girder section input, update the dimension of the original girder section with the average remaining dimension, for each subsequent section loss entry for that cross section it will use the newly updated dimension to compute the new average dimension.

3.10.2. Flange Section Loss

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Sample Position		CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Thickness	Dimension Token	Loss Type = SL: Leave Blank Loss Type = LT: Enter the thickness that was lost within the width indicated. Loss Type = RT: Enter the remaining thickness within the width indicated. Loss Type = LW: Enter the thickness the width loss applies to. Loss Type RW: Enter the thickness the remaining width loss applies to. Users can use Tokens instead of providing the dimension, see Note 2.
Width	Dimension Token	Loss Type = SL: Leave Blank Loss Type = LT: Enter the width that was lost within the thickness indicated. Loss Type = RT: Enter the remaining width within the thickness indicated. Loss Type = LW: Enter the width the thickness loss applies to. Loss Type RW: Enter the width the remaining thickness loss applies to. Users can use Tokens instead of providing the dimension, see Note 1.
Percent Thickness Loss	Percentage	Loss Type = SL: Enter the thickness percent section loss All other Loss Types = Leave blank.
Percent Width Loss	Percentage	Loss Type = SL: Enter the width percent section loss All other Loss Types = Leave blank.
Loss Type	LW LT RW RT SL	The following key words indicate the type of section loss input: LW = Entered dimensions are parameters to determine the lost width of the flange. LT = Entered dimensions are parameters to determine the lost thickness of the flange. RW = Entered dimensions are parameters to determine the remaining width of the flange. RT = Entered dimensions are parameters to determine the remaining thickness of the flange. SL = Section Loss Percentage

Notes:

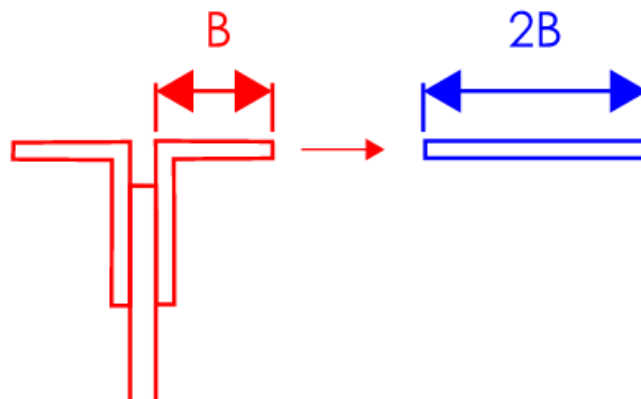
- Width tokens are used to automatically generate the width input based on previous inputs. Note that when overlapping ranges occur, the tokens will use the updated flange width from any previous section loss entries to determine the values as it loops through the section loss inputs for a given cross-section.

Token	Expansion	Remarks
FW	Full Width	The width dimension at runtime is replaced with the flange width, including any previous width losses for overlapping ranges. For angle flanges, this is the sum of both horizontal legs (2B shown in Note 3.)
FL	Full Leg	The width dimension at runtime is replaced with half the flange width, including any previous width losses for overlapping ranges. For angle flanges, this is the width of a single horizontal leg (B shown in Note 3.)

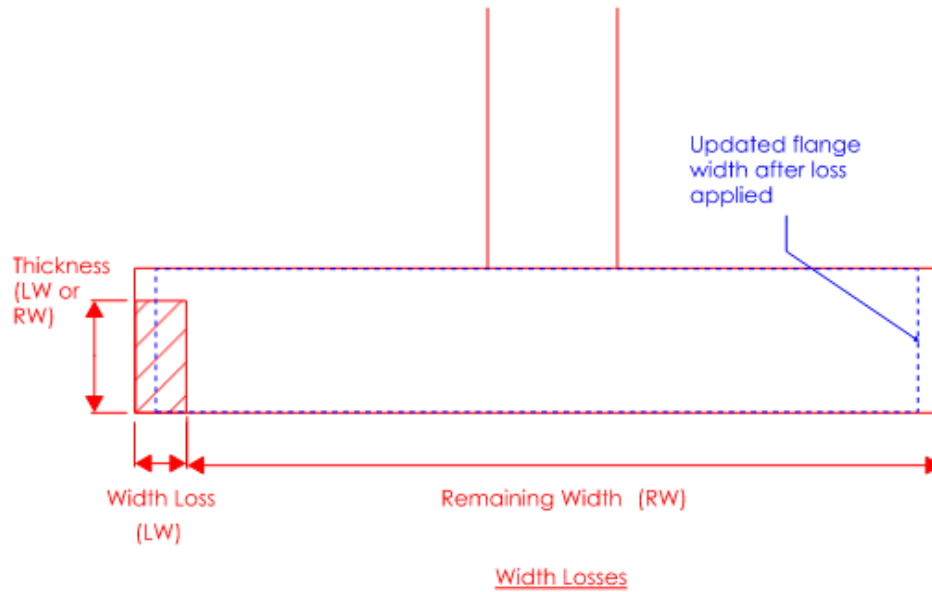
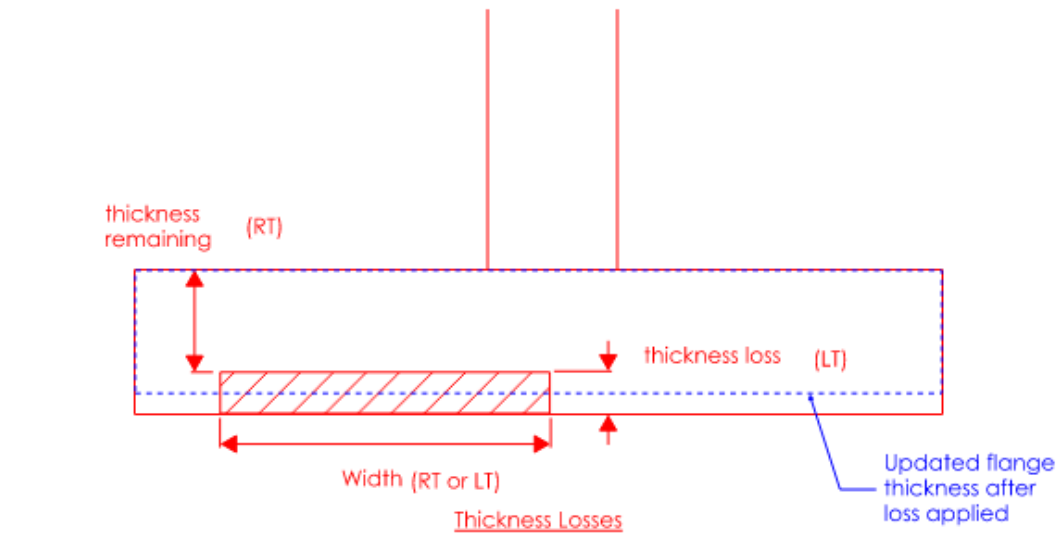
- Thickness tokens are used to automatically generate the thickness input based on previous inputs. Note that when overlapping ranges occur, the tokens will use the updated flange thickness from any previous section loss entries to determine the values as it loops through the section loss inputs for a given cross-section.

Token	Expansion	Remarks
FT	Full thickness	The thickness dimension at runtime is replaced with the flange thickness, including any previous thickness losses for overlapping ranges.
KE	Knife Edge	The width dimension at runtime is replaced with half the flange thickness, including any previous thickness losses for overlapping ranges.

- Users have the ability to determine if the section losses reduce the flange width dimension or flange thickness dimension for each row entry.
- Angle Flanges: Enter the losses with respect to both flanges. I.e., for loss types other than 'SL' enter the actual dimension of the loss; for 'SL' loss types, use the actual section loss area divide the sum of both flanges widths times the thickness. The vertical legs are not adjusted by the section loss entry, only the horizontal leg width should be accounted for in the section loss percentage calculations.



- Schematics:



3.10.3. Cover Plate Section Loss

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Sample Position		CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Thickness	Dimension Token	Loss Type = SL: Leave Blank Loss Type = LT: Enter the thickness that was lost within the width indicated. Loss Type = RT: Enter the remaining thickness within the width indicated. Loss Type = LW: Enter the thickness the width loss applies to. Loss Type RW: Enter the thickness the remaining width loss applies to. Users can use Tokens instead of providing the dimension, see Note 2.
Width	Dimension Token	Loss Type = SL: Leave Blank Loss Type = LT: Enter the width that was lost within the thickness indicated. Loss Type = RT: Enter the remaining width within the thickness indicated. Loss Type = LW: Enter the width the thickness loss applies to. Loss Type RW: Enter the width the remaining thickness loss applies to. Users can use Tokens instead of providing the dimension, see Note 1.
Percent Thickness Loss	Percentage	Loss Type = SL: Enter the thickness percent section loss All other Loss Types = Leave blank.
Percent Width Loss	Percentage	Loss Type = SL: Enter the width percent section loss All other Loss Types = Leave blank.
Loss Type	LW LT RW RT SL	The following key words indicate the type of section loss input: LW = Entered dimensions are parameters to determine the lost width of the flange. LT = Entered dimensions are parameters to determine the lost thickness of the flange. RW = Entered dimensions are parameters to determine the remaining width of the flange. RT = Entered dimensions are parameters to determine the remaining thickness of the flange. SL = Section Loss Percentage

Notes:

1. Previous section notes for the Flange apply to the cover plates. Except Angles are not applicable.
2. Section losses are always applied to the outermost cover plate.

3.11. Bracing

Point Bracing & Discrete Bracing is used to define discrete brace points along the member. Note that discrete brace points (e.g. diagram locations) can be entered in either "PointBracing" or "DiscreteBracing". It is recommended to use "PointBracing". "DiscreteBracing" does not have the versatility to define independent bracing of the top and bottom flange, and the input method is more cumbersome. The "PointBracing" input is intended to replace "DiscreteBracing". "DiscreteBracing" is still contained within the Member Input Book to retain compatibility with existing bridges in the database.

Note: Both the top and bottom flanges at each support location will be considered to be a brace point, regardless of entry in this table.

3.11.1. Point Bracing

Enter rows in this table to indicate brace point locations. Top and Bottom flanges may or may not be braced at the same location.

Parameter	Input Option	Remarks
Station	Station	Enter the station along the girder of the brace point
Top Flange	Boolean (TRUE or FALSE)	TRUE = Top flange is braced at this point FALSE = Top flange is not braced at this point
Bottom Flange	Boolean (TRUE or FALSE)	TRUE = Bottom flange is braced at this point FALSE = Bottom flange is braced at this point

3.11.2. Discrete Bracing

See Bracing Note above. Enter the unbraced ranges. Each range indicates an unbraced segment of both top and bottom flanges.

Parameter	Input Option	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.

3.11.3. Lateral Bracing

The top flange of any section within the range entered in this table will be taken as continuously braced. Any gaps of the member along not defined in this table we be taken as not continuously braced.

Parameter	Input Option	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.

Note: Top flanges of composite sections will be considered continuously braced. Entry is redundant and not required for such ranges.

3.12. Stiffeners

Member Book

Stiffeners

This sheet group contains the user inputs to apply transverse and longitudinal stiffeners to the girder web and define stiffener definitions.

3.12.1. Transverse Stiffeners

Member Book

Stiffeners

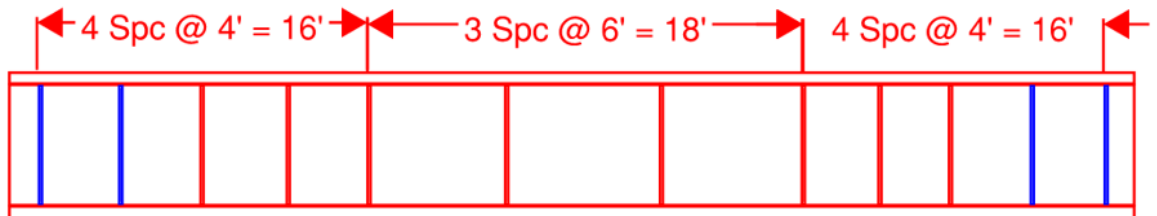
TransverseStiffener

Transverse stiffener ranges are entered by the user, instead of actual transverse stiffener locations. The user will specify a range and enter the stiffener properties within that range. The end stiffener definition is used when stiffener changes at the end of the range. The below images provide an example to populate the entries. Please note, transverse stiffeners will not appear in the schematic, as actual stiffener locations are not entered in the program.

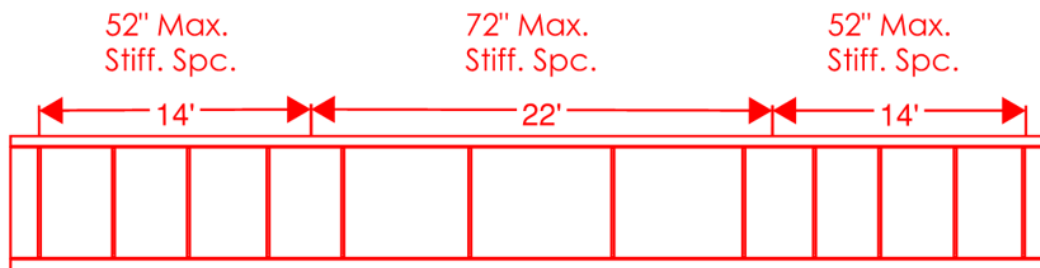
Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Spacing	Distance	Enter the stiffener spacing in this range
Number of Spaces	Integer	Not used/not required. Can be helpful in writing equations for the spacing input.
Stiffener Definition	Name	Enter the name of the Stiffener Definition inputted on StiffenerDefinition sheet exactly as it appears. The properties of this stiffener will be assumed for all sections within the entry range, except when a end stiffener definition is assigned, see below.
End Stiffener Definition	Name	Enter the name of the Stiffener Definition inputted on StiffenerDefinition sheet exactly as it appears. When not left blank, the properties of the specified end stiffener will be used for all section within the spacing input from the end of the range.

Examples:

Transverse Stiffener Spacing						
Start Distance (ft)	Length (ft)	End Distance (ft)	Spacing (in)	Number of Spaces	Stiffener Definition	End Stiffener Definition
0		8	48		Type Blue	Type Red
8		16	48		Type Red	Type Red
16	18	34	72		Type Red	Type Red
34	12	46	48		Type Red	Type Blue
46	4	50	48		Type Blue	Type Blue



Transverse Stiffener Spacing						
Start Distance (ft)	Length (ft)	End Distance (ft)	Spacing (in)	Number of Spaces	Stiffener Definition	End Stiffener Definition
0		14	52		Type Red	
14	22	36	75		Type Red	
36		50	52		Type Red	



3.12.2. Transverse Stiffener Definitions



Parameter	Input Options	Remarks
Stiffener Definition Name	Name	Enter the name for the stiffener. Must be unique.
Projecting Width	Dimension	Enter the projecting width of the stiffener. Do not include any fill plates in this dimension.
Thickness	Dimension	Enter the stiffener thickness.
isPair	Boolean (TRUE or FALSE)	TRUE = Stiffener pair FALSE = Single Sided Stiffener
Gap at Top	Dimension	Enter the gap between the stiffener and inside of top flange. May be left blank if no gap is present.
Gap at Bottom	Dimension	Enter the gap between the stiffener and inside of bottom flange. May be left blank if no gap is present.
Steel	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the names of the material entered on the Materials > Steel sheet.
Horizontal Leg Width	Dimension	For plate stiffeners leave blank. For angle stiffeners enter the leg width attached to the web.
Override	It=[value]	If the user would like to override the moment of inertia for the stiffener, enter It=value. Replace value with the moment of inertia of the stiffener to use in the analysis.
Fill Plate Thickness	dimension	If a fill plate is present, enter the thickness of the fill plate.

Examples:

Transverse Stiffener Definitions									
Stiffener Definition Name	Projecting Width (in)	Thickness (in)	isPair (boolean)	Gap at Top (in)	Gap at Bottom (in)	Steel	Horizontal Leg Width (in)	Override	Fill Plate Thickness (in)
Type Red	8	0.75	TRUE	0	0	A36			
Type Blue	8	0.75	FALSE	0	0	A36			
Type Green	8	0.75	FALSE	0	0	A36	4		0.5

where:

bt = projecting width

ts = thickness

HI = horizontal leg thickness

tf = fill plate thickness

gt = gap at top

gv = gap at bottom

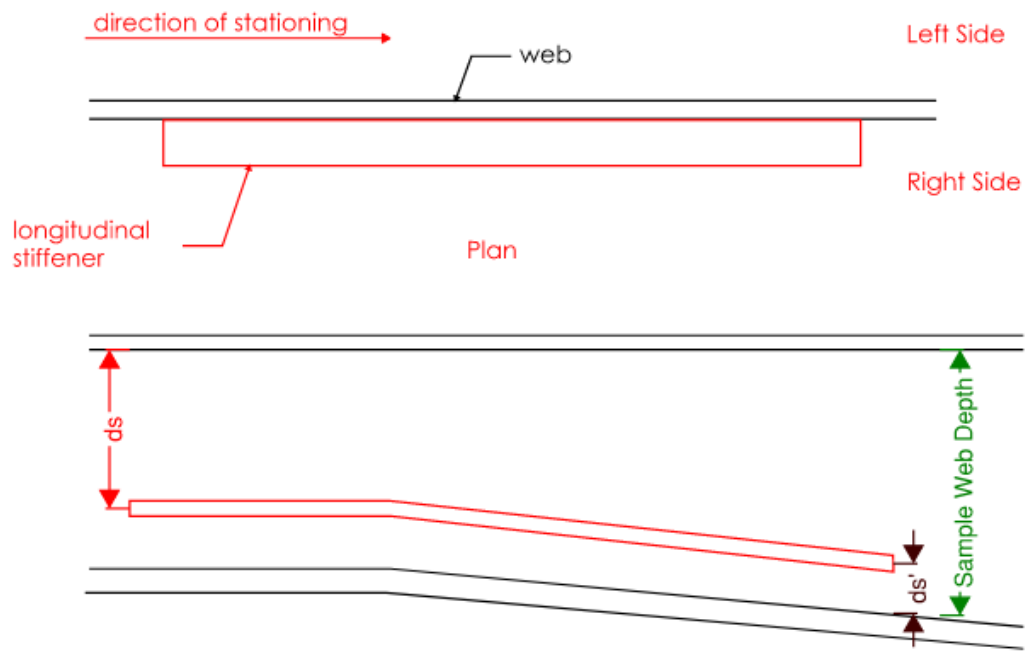
3.12.3. Longitudinal Stiffeners

Enter the station range and properties of each longitudinal stiffener on the girder.

Parameter	Input Options	Remarks
Start Station	Station	Enter the start station for the range.
Length	Distance	Not used. Users can use this cell to write formulas for start and end stations to aide in input.
End Station	Station	Enter the end station for the range.
Sample Position		CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Depth from top of web	Dimension	Enter the depth from the top of the web to location of the stiffener. Note, for stiffeners placed at the top of the web, it is usually more convenient to write an equation for the depth as a function of the Sample Web Depth cell.
Definition	Name	Enter the name of the Stiffener Definition inputted on LStiffenerDefinition sheet exactly as it appears.
Side of Web	Left Right	Indicate whether the stiffener is attached to the left side or right side of the web. Necessary for curved girders, doesn't hurt to apply for straight girders.
Sample Web Depth		CT-Steel will enter the web depth of each POI in this cell. See Article 3.1.1 Variable Parameter Cells

Note:

The Depth from top of web can be written as a function of the sample position and sample web depth cells. CT-Steel during runtime, the Engine will enter in the station of the POI into the sample position and depth of the web in the sample web depth cells before reading any other value. This is useful when the depth of the longitudinal stiffener varies or when construction records dimension the stiffener depth from the bottom flange.



Where:
 ds = Depth from top of web

note:
 sample web depth is entered by CT-Steel during runtime. The user could write an equation for ds where ds is a function sample web depth, useful for situations where the stiffener depth is dimensioned (ds') from the bottom flange on construction records.

3.12.4. Longitudinal Stiffener Definition

Parameter	Input Option	Remarks
Stiffener Definition Name	Name	Enter the name of the longitudinal stiffener. Name must be unique.
Projecting Width	Dimension	End the projecting width of the longitudinal stiffener.
Thickness	Dimension	Enter the thickness of the longitudinal stiffener.
Steel	Name	Enter the Name of the steel material for this range. The name shall match exactly one of the names of the material entered on the Materials > Steel sheet.
Longitudinal Stiffener Effectiveness	Decimal	Optional input. Used to apply an interpolation factor to the web load shedding factor between the web without a longitudinal stiffener and a web with a longitudinal stiffener. See note.
Include in Section Properties	Boolean	TRUE: Longitudinal Stiffener is included in elastic and plastic section properties. FALSE: Longitudinal stiffener is not included in elastic nor plastic section properties. Note if left blank this control will default to false.
Stage	Integer	If the stiffener is to be included in the section properties, indicate the starting stage it should be considered effective.

Note: The below equation is used by CT-Steel. This feature was original added for discontinuous longitudinal stiffeners.

$$R_b = R_{b,u}(1 - \eta) + \eta * R_{b,s}$$

Where:

- R_b = Web loading shedding factor used in rating calculations
- $R_{b,u}$ = Web load shedding factor based on unstiffened web
- $R_{b,s}$ = Web load shedding factor based on stiffened web
- η = Effectiveness factor entered by the user.

3.13. Fatigue

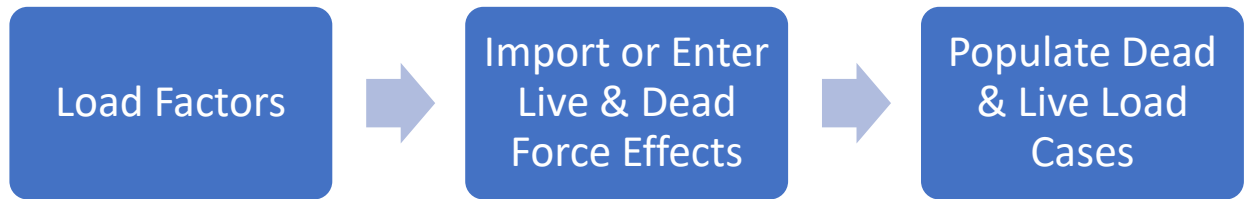
Each row in the this table can a single point where a fatigue detail applies or a range where the fatigue details apply.

Parameter	Input Option	Remarks
Detail Name	Name	Enter the name of the detail. E.g., shear connector, cover plate termination weld.
X, location	Station	Single Point Entry: Enter the station where the fatigue detail exists. Range Entry: Enter the start station of the fatigue detail.
Left or Right	Left Right Both Station	Single Point Entry: Enter whether the fatigue detail is at the left side or right side of the POI. An entry of Both will apply the fatigue detail to the left and right side of the station. Range Entry: Entering a numeric value will define the end station of a range input.
Fatigue Category	Letter	Enter the AASHTO fatigue category.
A, Detail Category Constant	Value	Enter the AASHTO Detail Category Constant, A
$(\Delta F)_{TH}$, Threshold	Value	Enter the AASHTO fatigue threshold.
Height from Bottom of Web	Dimension	Enter the depth from the bottom of the web. Note, negative values can be applied to section below the bottom of the web, such as the underside of flanges. Write a formula in this cell referencing the sample web depth to get a reference point to the top of the web, center of web, etc.
R_R	Value	AASHTO MBE Resistance Factor for the desired Evaluation Level.
Detail Year	Year	Year the detail was first introduced.
n	value	Estimated number of cycles per truck passage for the fatigue detail.
R_s	Value	AASHTO MBE Stress-Range Estimate Partial Load Factor
Sample Position		CT-Steel will enter the station of each POI in this cell. See Article 3.1.1 Variable Parameter Cells
Sample Web Depth		CT-Steel will enter the web depth of each POI in this cell. See Article 3.1.1 Variable Parameter Cells

3.14. Loads



The workflow for entering Load Cases and Force Effects in the Member Input book is as follows:



3.14.1. Load Factors

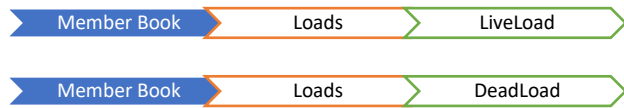


In the *LoadFactors* sheet review the Load Combinations for the dead and live load combinations the LRE plans to include in the rating. The LRE can add additional dead (permanent) and live load combinations to the table, with the corresponding load types and load factors.

Notes:

- Additional dead and live load combinations can be added to the end of the table, e.g., EH, EV, paving equipment, etc.
- The table contains the typical vehicles and load combinations, the LRE will need to review the load factors and update with using the member's actual load factors. The LRE does not need to delete load combinations that are not used for that specific member/bridge. Meaning, it is okay if a load combination is not used.
- Load Combinations do not need to be broken up by construction stage (non-composite, long-term, or short-term). The construction stage will be assigned to the load case, see Load Cases setup below.
- For Load Type enter "Dead Load" for all permanent load cases, and "Design", "Legal", "Permit", or "Emergency" as applicable for the live load combination.
- The Adjacent Vehicle load factor will apply the load factor to adjacent vehicle when adjacent vehicle load is applied in the Live Load force effects input.
 - Positive Load Factors for the adjacent vehicle will apply the adjacent vehicle demand in the numerator of the rating factor equation.
 - Negative Load Factors for the adjacent vehicle will apply the adjacent vehicle demand in the denominator of the rating factor equation. Note the absolute value of the load factor will be taken in the actual calculation.

3.14.2. Force Effect Input



3.14.2.1. Manual Entry

The LRE may manually enter the force effects, e.g., copying data from a FEA program. The LRE shall provide enough positions along the member to develop moment and shear diagrams. CT-Steel will read the moment and shear diagrams to determine force effects at any position through linear interpretation.

- **Position X:** is station of the member. The member stationing is defined at:



- **L/R:** The entry defines if that row is located at the left side or right side of the Position X. Leaving this cell blank, CT-Steel will assume that the force effects applies to both the left and right of the Position X. For example, at the interior support of a continuous member, the shear force effect to the left and right of the support may be different.
- **Load Case:** Enter the load case of the force effect. Note that the load type, e.g., DC, DW, HL-93, etc., will assigned to the load case, e.g., self-weight, wearing surface, HL-93 max envelope, etc., on the load cases sheet.
- **My** = Major Axis moment (kip-ft).
- **Fz** = Major Axis shear in (kip).
- **Mz top flange** = flange lateral bending moment of the top flange (without second order amplification, CT-Steel will amplify the flange lateral bending moment during the analysis) (kip-ft).
- **Mz btm flange** = flange later bending moment of the bottom flange (without second order amplification, CT-Steel will amplify the flange lateral bending moment during the analysis) (kip-ft).
- **Note:** the remaining columns are obsolete inputs and are not used by the engine.

Notes:

- Enter all force effects as Unfactored loads.
- CT-Steel will linearly interpolate loads for each POI (analysis section). The more positions along the span the LRE enters, the more accurate the rating will be.
- CT-Steel will sum the Force Effects at the same section which have the same load case. Therefore, ensure that duplicate force effects are not entered.
- Individual dead load cases should be entered that are broken up by construction stage (non-composite, long-term), e.g. DC1, and DC2.
- Individual load cases of a load combinations can be inputted separately, or combined. This means the LRE can enter "Girder Selfweight" and "Deck Weight" as separate entries into the table, or the LRE can enter a load case called "DC1" that has the girder and deck weight already summed.

- Vehicle force effects should be broken up in multiple load cases which maximize/minimize the force effect of concern. CT-Steel will rate each of these load cases separately.

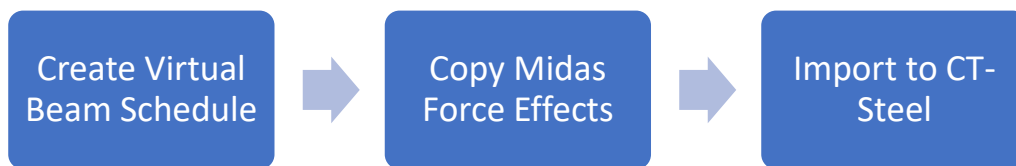
Load Case	
CT-L3S2 + Lane G1 FZmax	
CT-L3S2 + Lane G1 FZmin	
CT-L3S2 + Lane G1 MYmax	
CT-L3S2 + Lane G1 MYmin	

3.14.2.2. Automatic Input

Notes:

- This section of this user manual for the automatic input of force effects is not completed. CT-Steel may have additional functionality not described herein.
- The automatic importing tool cannot import flange lateral bending moments. The user will have to manually enter the flange lateral bending moments for bridge requiring consideration of the flange lateral bending stress in the load rating., except of LARSA, The LARSA importer tool can import flange lateral bending moments.

3.14.2.2.1. Midas Virtual Beams



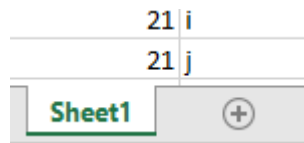
Virtual Beam Schedule

The LRE shall create a virtual beam schedule which relates the "i" or "j" the Stationing of the member, as defined in:



The Virtual Beam Schedule shall be a excel workbook. On "Sheet1" create the columns starting.

	A	B	C	D	E
1					
2		Virtual Beam	Part	Sta	
3		1	i	0	
4		1	j	3.2	
5		2	i	3.2	
6		2	j	6.4	
7		3	i	6.4	
8		3	j	9.6	
9		4	i	9.6	
10		4	j	12.8	
11		5	i	12.8	



Note it is critical that the table starts in Cell B2 and the worksheet is named "Sheet1", as shown above.

Enter the station of the virtual beam in column D. It is encouraged to use formals, vlookups, etc. to generate the station list.

Copy Virtual Beam Force Effects

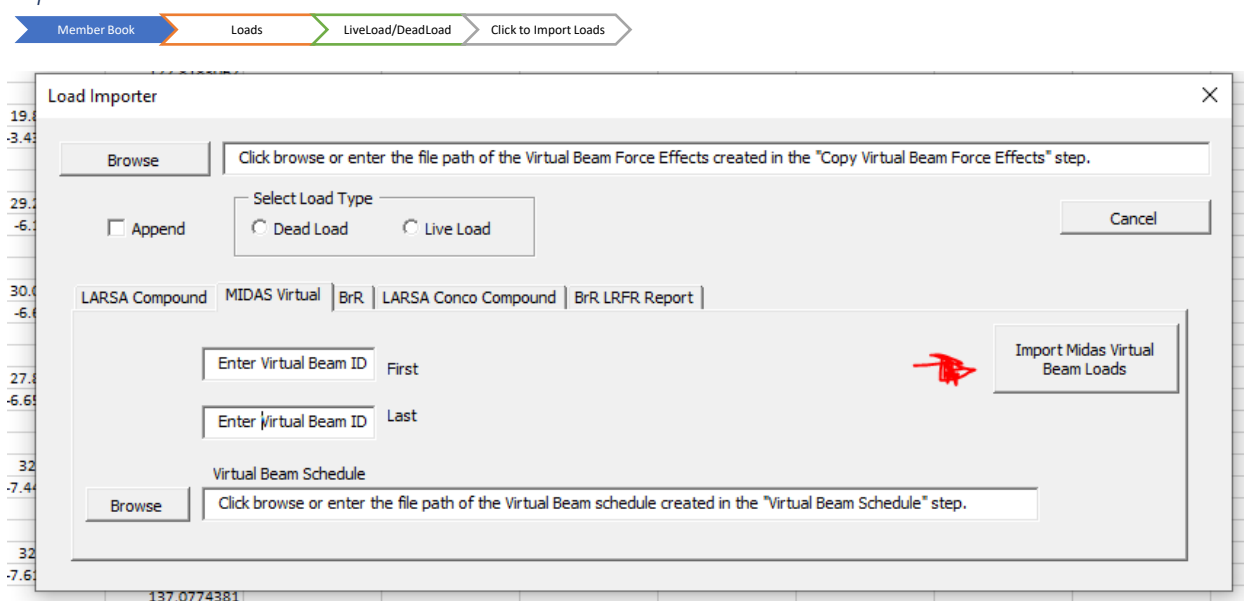
In a separate workbook copy and paste the virtual beam results from Midas into cell "A1" on "Sheet1".

	A	B	C	D	E	F	G	H	I	J	K	L	M
1													
2		Virtual Be Load		Stage	Step	Part	Axial (kip)	Shear-Y (k	Shear-Z (k	Torsion (in	Moment-Y	Moment-Z (in*kips)	
3		1 DC1	Final	001(last)	I		0.14	0.09	-54.3	-18.88	-0.18	8.31	
4		1 DC1	Final	001(last)	J		0.14	0.09	-48.7	-405.3	1939.03	3.82	

Note:

- The dead and live load force effects shall be copied into separate workbooks.
- Before copying the loads from midas, set the unit system to "in." and "kip". Note that CT-Steel will convert inches to feet. The LRE shall check the import for proper unit conversion.
- Note that all of the virtual beams can be copied into one workbook. In the next step, the user will have the option to select only the virtual beams which correlates to that member.

Import Into CT-Steel



Steps (performed separately for dead and live loads):

1. Ensure stationing has been defined for the member:



2. Click to Import the loads
3. Click "Midas Virtual" tab in the *Load Importer* tool window.
4. Click browse and select the dead or live load workbook that was created in the "Copy Virtual Beam Force Effects".
5. Select the Load Type radio button.
6. By default the importer will remove all previous dead or live load as applicable when then enter the new loads. If the LRE wishes to add additional load cases to an existing member input book, the LRE should check *Append*.
7. Use the *First* and *Last* input fields to filter the virtual beams which correspond to the member input book. Meaning, if the force effects workbooks has the virtual beams for girders G1 thru G10,

and the LRE is working on G3. The LRE would enter first virtual beam of G3 in the First field, and the last virtual beam on G3 in the Last field. The Load Importer tool will only import the virtual beams for that are within that range (including the first and last virtual beams).

8. In the lower input box, use Browse to enter the file path for the virtual beam schedule created in "Virtual Beam Schedule" step.
9. Click "Import Midas Virtual Beam Loads". (Note this may take more time than expected to generate).

3.14.2.2.2. Larsa Concomitant Compound Element Forces

This tool will import the Larsa Concomitant Compound Element Forces into the member input book. Concomitant Compound Element Forces are typically used to capture the coincidental flange lateral bending force with the major axis bending moment of the composite section from a 3D structural analysis.

A template input book to extract the loads from Larsa can be found in the STL-AppTools folder: LARSA-CCEFE.xlsm

Steps:

1. Extract the CCEFE from Larsa. Only one girder line shall be included in the LARSA-CCEFE spreadsheet. Dead and Live loads cannot be included in the same workbook (i.e., all dead load cases can be included in one spreadsheet but it cannot contain live loads).
2. The bridge path coordinate system and stationing in the Larsa extraction shall be the same as the stationing in the member input book.

Load Importer

Browse LARSA-CCEFE Dead Load File Path.xlsm

Append

Select Load Type

Dead Load Live Load

Cancel

LARSA Compound MIDAS Virtual BrR LARSA Conco Compound BrR LRFR Report

Shear Axis

Fx Fy Fz

Major Moment Axis

Mx My Mz

Lateral Moment Axis

Mx My Mz

Import Enveloped Force Effects Only

Import LARSA Compound Element Forces

Moment Scale Factor:

Lateral Scale Factor:

Shear Scale Factor:

Load Importer

×

Browse

LARSA-CCEFE Live\Load File Path.xlsm

☐ Append

Select Load Type

☐ Dead Load

☒ Live Load

Cancel

LARSA Compound

MIDAS Virtual

BrR

LARSA Conco Compound

BrR LRFR Report

Shear Axis

☐ Fx

☐ Fy

☒ Fz

Major Moment Axis

☐ Mx

☒ My

☐ Mz

Lateral Moment Axis

☐ Mx

☐ My

☒ Mz

☒ Import Enveloped Force Effects Only

Import LARSA Compound Element Forces

Moment Scale Factor:

Lateral Scale Factor:

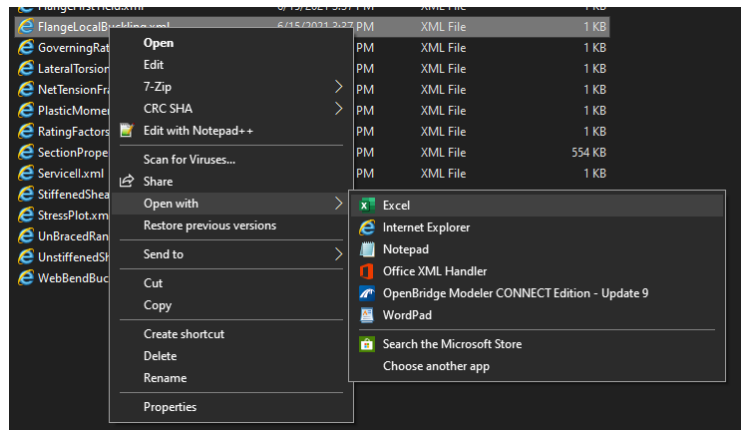
Shear Scale Factor:

4. Specification Checks

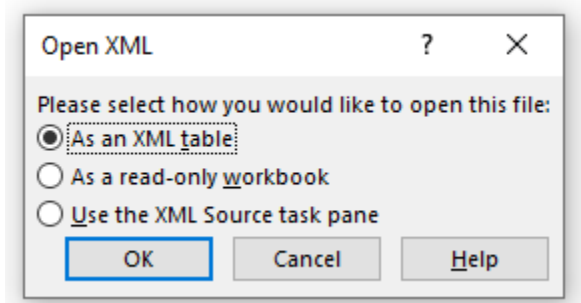
4.1. General

Specification outputs are outputted as xml files. To review the output, it is recommended to import the xml file to a blank excel workbook. Note that the excel workbook will need to be closed before attempted to run CT-Steel to avoid file access issues.

1. Open with excel from the file explorer. Note do not set excel as the default application for xml files. This will prevent proper display of the Results reports (Input Review, Overall Rating Summary) and will also affect the proper display of BrR reports.



2. Open as an xml table



4.2. Rating Failure Mode Articles

The subsequent sections define the output variables shown in the SpecCheck articles.

Each rating article contains the following common headings:

Heading	Unit	Description
Event		Rating Event ID
x	feet	Station of Point of Interest
SIDE		left section or right section of Point of Interest
PSPN		Parent Span Name
SPN		Span Name
PCT		Percent of Span expressed as a decimal
VEH		Vehicle Name
LC		Live Load Case Name
PROC		Rating Procedure
LS		Limit State
RF		Rating Factor

4.2.1. FlangeYielding

This article rates tension flanges and continuously braced compression flanges for non-compact sections using BDS Article 6.10.7.2, 6.10.8.1.2, 6.10.8.1.3, A6.1.2, A6.1.3, A6.1.4.

Heading	Unit	Description
FNG		Flange ID: TF – Top Flange BF – Bottom Flange TCP# – Top Cover Plate BCP# - Bottom Cover Plate Where the # is the position of the cover plate starting from the web.
C	ksi – for 6.10 articles Kip-in for A6 articles	Capacity used in the Rating Factor Equation
Fnt	ksi	Tension flange nominal resistance
Rh		Hybrid factor
Fyt	ksi	Yield Strength of tension flange
Myt	Kip-in	Yield moment to the tension flange
Mnt	Kip-in	Nominal moment resistance based on the tension flange
Sxt	In ³	Elastic section modulus to the tension flange at first yield
Rb		Web Load Shedding Factor
Fyc	ksi	Yield strength of the compression flange
Fnc	ksi	Nominal resistance of the compression flange
Rpc		
Sxc	In ³	Elastic section modulus to the compression flange at first yield
Myc	Kip-in	Yield moment to the compression flange
Mnc	Kip-in	Nominal moment resistance based on the compression flange
f_DL	ksi	Factored Dead load stress at the flange of interest
f_AL	ksi	Factored Adjacent vehicle stress at the flange of interest
f_LL	ksi	Factored Live load stress at the flange of interest
fl_DL	ksi	Factored Flange lateral bending stress due to dead load at the flange of interest
fl_AL	ksi	Factored Flange lateral bending stress due to adjacent vehicle at the flange of interest
fl_LL	ksi	Factored Flange lateral bending stress due to live load at the flange of interest.
M_DL	Kip-in	Factored dead load moment
M_AL	Kip-in	Factored adjacent vehicle moment
M_LL	Kip-in	Factored live load moment

4.2.2. Flange Local Buckling

This article contains rating output for Flange Local Buckling using Article 6.10.8.2.2 and A6.3.2.

Header	Unit	Description
FNG		Flange ID: TF – Top Flange BF – Bottom Flange TCP# – Top Cover Plate BCP# - Bottom Cover Plate When the # is the position of the cover plate starting from the web.
C		Capacity used in the rating factor equation
Mnc		Nominal moment resistance for local buckling
Fnc		Nominal
Fyc		
Fyr		
b	in	Width used to determine slenderness
t	in	Thickness used to determine slenderness
lamda_f		Slenderness ratio for the compression flange
lamda_pf		Limiting slenderness ratio for a compact flange
lamda_rf		Limiting slenderness ratio for the noncompact flange
E		Modulus of elasticity
Fyw		Yield strength of the web
Rpc		
Sxc		Elastic section modulus to the compression flange at first yield
Myc		Yield moment
Rh		Hybrid factor
Dc		Depth of web in compression
Rb		Web-load-shedding factor
f_DL	ksi	Factored Dead load stress at the flange of interest
f_AL	ksi	Factored Adjacent vehicle stress at the flange of interest
f_LL	ksi	Factored Live load stress at the flange of interest
fl_DL	ksi	Factored Flange lateral bending stress due to dead load at the flange of interest
fl_AL	ksi	Factored Flange lateral bending stress due to adjacent vehicle at the flange of interest
fl_LL	ksi	Factored Flange lateral bending stress due to live load at the flange of interest.
M_DL	Kip-in	Factored dead load moment
M_AL	Kip-in	Factored adjacent vehicle moment
M_LL	Kip-in	Factored live load moment

4.2.3. Lateral Torsional Buckling

Strength – Flexure

Scope: This spec article includes lateral torsional buckling data ratings prepared using Section 6.10.8 & A6.

Heading	Unit	Description
UBR		Unbraced Range key of the flange under consideration.
FNG		Flange under consideration, bottom or top.
C	ksi or Kip-in	Capacity used in the rating factor equation.
Fnc	ksi	Nominal compressive resistance based on LTB. Note, when A6 is used, Fnc is outputted as Mnc/Sxc, for information only.
Rb		Web Load Shedding Factor. See 'WebLoadSheddingFactor' specArticle for additional parameters used to determine this factor.
Rh		Hybrid Factor
Fyc	ksi	Yield strength of compression flange
Fyr	ksi	Compression flange stress on the on-set of nominal yielding.
Fe	ksi	Elastic Latral-Torsional Buckling Stress
Cb		Moment Gradient Modifier
Lb	in	Unbraced Length
Lp	in	limiting unbraced length to achieve the nominal flexural resistance of $R_b R_h F_{yc}$.
Lr	in	limiting unbraced length to achieve the onset of nominal yield in either flange under uniform bending considering compression-flange residual stress and geometric imperfection effects.
Sxc	in ³	Elastic section modulus to the compression flange at first yield.
Phi		Flexural resistance factor
fbu_LL	ksi	Factored Live load stress taken at the governing section.
fbu_DL	ksi	Factored dead load stress taken at the governing section.
fbu_AL	ksi	Factored adjacent vehicle load stress taken at the governing section.
i		iteration for converging on rating factor.
Dc	in	Depth of Web in Compression
rt	in	effective radius of gyration for lateral-torsion buckling.
MA	kip-in	Factored Moment taken at first-quarter point of the unbraced length
MB	kip-in	Factored Moment taken at mid-point of the unbraced length
MC	kip-in	Factored Moment taken at third-quarter point of the unbraced length
Me1A	kip-in	Minimum elastic lateral-torsion buckling moment within the tributary portion to the first-quarter point of the unbraced length.
Me1B	kip-in	Minimum elastic lateral-torsion buckling moment within the tributary portion to the mid-point of the unbraced length.
Me1C	kip-in	Minimum elastic lateral-torsion buckling moment within the tributary portion to the third-quarter point of the unbraced length.
Chi		non-prismatic geometry modification factor
Cbe		Effective Moment Gradient Factor for non-prismatic unbraced lengths.

MuMe1_max		Maximum ration of Mu/Me1 within the unbraced length
Gamma_e		Elastic lateral-torsional buckling load ratio

4.2.4. Net Tension Fracture

Header	Unit	Description
FNG		Flange ID: TF – Top Flange BF – Bottom Flange TCP# – Top Cover Plate BCP# - Bottom Cover Plate When the # is the position of the cover plate starting from the web.
C	ksi	Capacity used in the rating factor equation
Fnt	ksi	Nominal resistance
An	in ²	Net area of flange
Ag	in ²	Gross Area of flange
Fyt	ksi	Yield strength of flange
Fut	ksi	Ultimate strength of flange
f_DL	ksi	Factored dead load stress
f_AL	ksi	Factored adjacent vehicle stress
f_LL	ksi	Factored live load stress

4.2.5. Plastic Moment

Heading	Unit	Description
C	Kip-in	Capacity used in the rating factor equation
Mn	Kip-in	Nominal moment capacity
Dp	in	Depth of web in compression
Dt	in	Depth of composite section
Rh		Hybrid Factor
Mp	Kip-in	Plastic Moment
My	Kip-in	Yield Moment
Sxt		Elastic section modulus to the tension flange
isDet		TRUE: Section is in a span segment which is statically determinate FALSE: Section is in a span segment which is not statically determinate
fl_DL	ksi	Factored Flange lateral bending stress due to dead load at the flange of interest
fl_AL	ksi	Factored Flange lateral bending stress due to adjacent vehicle at the flange of interest
fl_LL	ksi	Factored Flange lateral bending stress due to live load at the flange of interest.
M_DL	Kip-in	Factored dead load moment
M_AL	Kip-in	Factored adjacent vehicle moment
M_LL	Kip-in	Factored live load moment

4.2.6. Stiffened Shear

This article contains rating output for transverse stiffened web panels.

Header	Unit	Description
C	kip	Shear capacity used in the rating factor equation
Vn	Kip	Nominal shear resistance
Vcr	Kip	Shear-yielding or shear-buckling resistance
Panel		End: End Panel Int: Interior Panel
Def		Transverse Stiffener definition for panel
Vp	Kip	Plastic Shear Force
D	in	Overall web depth
tw	in	Web thickness
K		Shear buckling coefficient
d0	in	Stiffener spacing
Cv		Ratio of the shear-buckling resistance to the shear yield strength
Fyw	Ksi	Yield Strength of web
E	ksi	Modulus of elasticity of web
VnPB	Kip	Nominal Post-Buckling Tension Field Action shear resistance
It	in ⁴	Moment of inertia of stiffener
It1	in ⁴	Minimum moment of inertia of the transverse stiffener required for the development of the web shear-buckling resistance
It2	in ⁴	Minimum moment of inertia of the transverse stiffener required for the development of the full web buckling plus postbuckling tension-field action resistance
rho_w		Ratio of maximum postbuckling resistance to the full postbuckling resistance
Aft	in ²	Area of top flange
Afb	in ²	Area of bottom flange

4.2.7. Unstiffened Shear

This Article contains ratings for unstiffened webs.

Header	Unit	Description
C	Kip	Shear strength used in rating factor equation
Vn	Kip	Nominal shear strength
K	Kip	Shear-buckling coefficient
d0	In	Stiffener spacing
Hc	in	Unsupported web height
D	In	Total web depth
Cv		Shear buckling to shear yielding ratio
Fyw	Ksi	Yield strength of web
tw	in	Web thickness
E	ksi	Modulus of elasticity of web

4.2.8. Service II

Strength Limit State - Flexure

Heading	Unit	Description
FNG		Flange ID: TF – Top Flange BF – Bottom Flange TCP# – Top Cover Plate BCP# - Bottom Cover Plate When the # is the position of the cover plate starting from the web.
C	ksi	
fr	Ksi	Allowable flange stress
Fyf	Ksi	Yield Strength of flange
Rh		Hybrid Factor
f_DL	Ksi	Factored dead load stress in flange extreme fiber
f_AL	Ksi	Factored adjacent vehicle live load stress in flange extreme fiber
f_LL	Ksi	Factored live load stress in flange extreme fiber
fl_DL	Ksi	Factored dead load lateral bending stress at the tip of the flange
fl_AL	Ksi	Factored adjacent vehicle live load lateral bending stress at the tip of the flange
fl_LL	Ksi	Factored live load lateral bending stress at the tip of the flange

4.2.9. Web Bend Buckle

Service II Limit State - Flexure

Header	Unit	Description
C	Ksi	Capacity used in the rating equation. $C = -F_{crw}$
F_{crw}	Ksi	Critical web bend buckling capacity.
k	Dim.	Bend-buckle coefficient
Hw	In	Effective depth of web
E	Ksi	Modulus of elasticity of web
f_DL	Ksi	Factored dead load stress at edge of web. (Negative values indicate compression)
f_AL	Ksi	Factored adjacent live load stress at edge of web (Negative values indicate compression)
f_LL	ksi	Factored live load stress at edge of web (Negative values indicate compression)
Dc	in	Depth of web in compression
Stiffened		Y = Web was considered longitudinally stiffened N = Web was not considered longitudinally stiffened

4.2.10. 1Fatigue

Fatigue Limit State - Flexure

Heading	Unit	Description
Detail		Echo of detail name.
Cat		Echo of detail fatigue category.
Y	inches	Distance from the bottom of the web to fatigue detail. I.e., elevation of fatigue details with the datum taken as the bottom of the web.
del_f_eff_II	ksi	Factored Effective fatigue stress range as the fatigue II limit state. Calculated as $0.8 * R_p * R_s * \Delta f$. MBE Eq. 7.2.2-1. Note 0.8 is the Fatigue II load factor.
del_f_eff_II_tension	ksi	Tension portion of the factored Effective fatigue stress range at the Fatigue II limit state. MBE variable: $(\Delta f)_{tension}$ in 7.2.3.
del_f_max		The maximum stress range expected at the fatigue-prone detail, taken as R_p times the factored calculated stress range for the fatigue I combination. Calculated as $1.75 * R_p * \Delta f$. MBE 7.2.4. Note 1.75 is the Fatigue I load factor.
f_DL	ksi	Unfactored dead load stress.
f_DL_compression	ksi	Unfactored dead load compressive stress.
Prone	Boolean	Is the detail fatigue prone per MBE 7.2.3.
Rp		Fatigue multiple presence factor
ADTT_Present	Trucks/day	Echo of present ADTT for all directions of truck traffic including all lanes on the bridge.
nL		Echo of number of lanes
L	ft	Span Length
f_max	ksi	Unfactored stress resulting from the fatigue vehicle load cases which produces the largest positive moment. Max refers to the moment envelope.
f_min	ksi	Unfactored stress resulting from the fatigue vehicle load cases which produces the largest negative moment.
F_th	ksi	Echo of fatigue threshold. Min refers to the moment envelope
ADTT_SL_PRESENT	Trucks/day	Echo of present ADTT in a single lane
ADTT_SL_Future	Trucks/day	Future ADTT in a single lane corresponding to Y_rem0.
g	decimal	Traffic growth rate
Y_rem0	years	Remaining life if $ADTT_SL_Future \leq ADTT_SL_Limit$. MBE Eq. 7.2.5.1-4.
Y_rem	years	Estimate remaining fatigue life.
Y_ADTT_Limit		Number of years from present day until the ADTT_SL_Limit is reached.
YearExh	Year	Estimate year for fatigue detail exhaustion.
age	Year	Age of fatigue detail.
Rr	Dim.	Echo of resistance factor.
A	(ksi) ³	Constant Amplitude Factor. Note: This value is entered by the user as 10^8 (ksi) ³ . The output here is expressed as 10^0 (ksi) ³ .
Nav	Cycles	Number of initially available fatigue stress cycles

N1	cycles	Number of fatigue cycles consumed over the present age of the detail.
n	Cycles	Echo of cycles per truck passage.

4.2.11. PlasticSectionProperties

This article contains generic plastic moment properties for each section.

Heading	Unit	Description
Stage		Stage
Bending		+: Positive Flexure -: Negative Flexure
Mp	Kip-in	Plastic moment capacity
Yp	In	Position of the plastic natural axis, measured from the bottom of the web.

5. Schematic

CT-Steel will generate a 3D schematic of the girder in CAD format (dxf) which is outputted into the SpecCheck folder for the member. A Microstation dgn file is copied into the SpecCheck folder and set to reference in the dxf file for viewing.

The following attributes are displayed in the schematic:

1. Girder Web, Flanges, Cover Plates, Side Plates
2. Deck Slab & Reinforcement*
3. Flange Brace Points
4. Locations of Simple Supports and Hinges
5. Support Locations
6. Longitudinal Stiffeners

*The width of the reinforcement shape is equal to the area of the reinforcement entered. E.g., reinforcement entered as 5 sq. in. will be drawn with a width of 5 inches.

Notes:

1. The vertical lines in the schematic represent analysis locations (POIs).
2. The schematic is generated during the "POI" or "Run" events. Loads are not required to be entered to generate the schematic.
3. Transverse stiffeners are not shown in the schematic because the exact location of the stiffeners are not entered by the user.

6. Updates

This section contains descriptions of updates starting from Version 2.10.

6.1. Version 2.10

6.1.1. Enhancements

1. Adjacent Vehicles in the Rating Factor Equation:
Users can control if the adjacent vehicle demand is applied in the numerator or denominator of the rating factor equation. There are two places where the user can control how the adjacent vehicle demand is applied in the rating factor equation.
 - a. Load Factor Sheet: Users can control the application of the adjacent vehicle by using a negative or positive load factor. This allows the user to control the application of the adjacent vehicle for each rating vehicle and limit state.
 - b. Control Options: Users can control how the adjacent vehicle demand is applied in the rating factor equations for all vehicles and limit states. Users can use this setting to defer to the entry on load factor sheet or override the entry on the load factor sheet.
2. Prismatic Checks:
 - a. New control option was added, isPrismaticTolerance, which allows the user to apply a tolerance applied to the check to determine if the member is prismatic within the unbraced range.
 - b. New control option was added, PrismaticRangeLimits, which allows the user to exclude sections at the end of the unbraced range with the smaller flange compressive stress when check to see if the member is prismatic within the unbraced range.
3. Moment Gradient Modifier Method:
A new option was added for users to select between the AASHTO 9th Edition Cb equations or the AISC equations for the moment gradient modifier.
4. Added sample position and sample web depth to fatigue input table.
5. Transverse stiffeners inputted with more the 6tw gap to the compression flange will not be considered stiffened.

6.1.2. Bug Fixes

1. Control Range Overrides:
Fixed bug where control option range overrides were not appropriately being established.
2. Appendix A6 LTB Ratings:
 - a. ~~The maximum moment in the unbraced range was previous back calculation from the maximum compressive stress within the unbraced range and the section modulus of the compressive flange. This approach could lead to higher moments when elements are added at later stages, such as cover plate or deck reinforcement.~~ Reverted in version vB2.10.2
 - b. Fixed calculation for F_{yr}
 - c. Fixed calculation for J
 - d. Fixed L_r in check for D6.4
3. Eligibility of Flanges for LTB Calculation:
Previously, flanges subject to live load tension stresses in the unbraced range where not considered eligible as the smallest resistance in the unbraced range when computing LTB rating factors. The update in this version will now consider such flanges are eligible as the smallest resistance within the unbraced range.

4. Fixed web height is determining slenderness of web when compute the shear buckling ratio when UseAISC_Shear_C_value control option is enabled.
5. Removed lateral bending effects from continuously braced tension flanges rating.
6. Fixed issue reading the 'Rs' value for fatigue from the user's input.
7. Replaced supported web depth with total web depth is used in BDS Eq. 6.10.9.3.2-1.

6.1.2.1. Hot Fixes

6.1.2.1.1. vB2.10.1

Fixed issue “Edit Member” button not opening the member input book.

6.1.2.1.2. vB2.10.2

Reverted a change made in vB2.10 which used the maximum moment in the unbraced range when computing LTB under Appendix A6. Section 4.6 of the Method to Solution describes to the current method.

6.2. Version 3

6.3. Version 3

To use version 3, select Version 3 using the radio button on the “Control” worksheet of CT-Steel.

Since the output location for the results has changed, users can run Version 3 without overriding results generated by Version 2, allowing users to compare results between the two versions.

6.3.1. Getting Started

Before running members in Version 3, select the desired members (check box in member list), then click the “Update” button on the Control sheet.



This function will migrate the member data to the latest input book. This will make available new control options and input fields required for Version 3.

6.3.1.1. Control Options

The following control options have been added:

- NonPrismaticMethod
- GammaE_MethodC
- UseMethodAForNonCompositeSinglySymReverseCurve
- ConsiderFatigueShear

6.3.1.2. Input Fields

- Bridge – Span – Boundaries
- Bridge – Span – Vertical Geometry
- Stiffeners – LstiffenerDefinition – Include in Section Properties
- Stiffeners – LstiffenerDefinition – Stage

6.3.2. SpecCheck Locations

The SpecCheck output for version 3 has been relocated to a sub-folder, named [STL-SpecCheck V3](#). This change will prevent version 3 runs from overriding existing version 2 runs.

Additionally, SpecChecks folders now use the file name of the bridge model as opposed to the entered member identification. Note: if the user changes the file name of the member after running, the user will need to rerun the member.

6.3.3. Reports Location

Version 3 Reports are now saved to a sub folder of the version 2 reports locations, named [Version 3](#).

6.3.4. Boundary Conditions

In version 2, users entered locations of simple supports and hinges. This input was used to set the boundary conditions of the span, to determine which sections were considered statically determinate and locations of the end panels for shear. For situations of cantilevers, users would use a workaround by setting the fixed end of the cantilever as a “hinge”, to force a statically indeterminate condition. This would also conservatively, mark the panels adjacent the fixed end of the cantilever as end panels. In

version 3, users can now specify locations hinges, simple supports, continuous supports, free ends, and fixed ends to better control the statically determinate condition without effecting the shear calculations.

For legacy models with only the locations of hinges or simple supports defined, CT-Steel, will assume any entered station under boundaries without a boundary type specified is simple support and the starts and ends of spans, without a station defined in the boundary conditions, is a continuous support. See User Manual for additional examples for entering the boundary condition data.

6.3.5. Vertical Geometry

A new input field for vertical geometry has been added to CT-Steel. The vertical geometry is required for members where variation in girder web depth is made in the top flange. Members which constant depth girders or members where the variation in web depth is made in the bottom flange, this input field may be left blank.

6.3.6. Methodology Changes

6.3.6.1. Lateral Torsional Buckling

Lateral Torsional Buckling and its associated parameters are computed in accordance with 2022 AASHTO Bridge Committee Agenda Item 30 (Revision 1). Method's A, B, C of the Ballot Items are fully supported. In the Lateral Torsional SpecCheck output, the notes column will indicate this by 10thB. (Note the B indicates it's a ballot item proposal for the 10th edition of the BDS).

The ballot items better addresses non-prismatic unbraced lengths. For prismatic unbraced lengths, changes were incorporated in the 10th edition ballot items, which can improve LTB rating factors.

The following control options have been disabled in version 3:

- CbMethod
- isPrismaticTolerance
- PrismaticRangeLimits

These control options may not be required based on the new LTB provisions. If these control options are needed to be reinstated, please Contact CTDOT.

6.3.6.2. Web-Bend Buckling Rating with Longitudinal Stiffeners

CT-Steel will check if the longitudinal stiffener meets the current AASHTO rigidity requirements and detailing to be considered effective for determining the optional Service II load rating criteria for web-bend buckling.

6.3.6.3. Sections with Multiple Longitudinal Stiffeners

Webs with multiple longitudinal stiffeners in its web depths, the most optimal stiffener will be used to determine web load shedding factor.

6.3.6.4. Structural Longitudinal Stiffeners

Users can now indicate if longitudinal stiffeners can be included in the section properties (e.g., moment of inertia and plastic moment capacity). Existing models will default to not consider the stiffener in the section properties. User's will have to enter "TRUE" for the Structural Stiffener input on the Longitudinal Stiffener Definition input. User's will have to determine if the longitudinal stiffener can be included in the section properties for the specific bridge.

6.3.6.5. *Fatigue Shear*

Added fatigue shear ratings for web subjected to post-buckling. Users can enable or disable this rating check with a control option.

6.3.7. *Progress Tracking*

The List Box of members will highlight the current member which it is rating. The status bar will show the progress for each member based on the number of rating events.