AASHTOWare BrDR 7.5.0 3D FEM Analysis Tutorial Mesh Generation and Dead Load Analysis Example

AASHTOWare Bridge Design and Rating Training 3DFEM5 – Mesh Generation and Dead Load Analysis Example

Topics Covered:

- 3D model deck mesh generation
- Dead load only analysis
- 3D model validation

Overview of mesh generation options:

- Number of shell elements
- Target aspect ratio for shell elements
- Node merge tolerance by span

This tutorial describes the data entry for a curved steel girder system bridge in BrDR version 7.5. This tutorial illustrates the process of using the dead load only analysis to determine appropriate mesh generation options for a particular model. The impact of each mesh generation option on the deck shell elements is discussed.

Example Curved Steel Girder Bridge

Framing Plan



Structure Typical Section



Curved Girder Structure Data Entry

Create a new bridge from the bridge explorer menu.

BRIDGE EXPLORER BRIDGE FOLDER RATE TOOLS VIEW	
New Open Description Bridge Manage	
→ Favorites Folder → P Recent Bridges BID Bridge ID	
All Bridges 1 TrainingBridge1 Tr	raining B
E Palated Bridges 2 TrainingBridge2 Tr	raining B
3 TrainingBridge3 Tr	raining B
4 PCITrainingBridge1 PC	Cl Trainir
5 PCITrainingBridge2 PC	CITrainin
6 PCITrainingBridge3 PC	CI Trainir
7 PCITrainingBridge4 PC	CITrainin
8 PCITrainingBridge5 PC	Cl Trainir
9 PCITrainingBridge6 PC	CITrainin

Assign the bridge ID, NBI structure ID, bridge name and other relevant information. Because this bridge will only contain a superstructure, only the 'Superstructures' option needs to be selected in the Bridge Workspace View. After inputting the bridge description, select 'OK' to create the new structure.

Mesh Generation	eration NBI structure ID (8): Mesh Generation	Template Bridge compl	etely defined	 − □ × Bridge Workspace View Superstructures Culverts Substructures
Description Desc	ription (cont'd) Alternatives Global reference point	t Traffic Custom agency fie	lds	
Name:	Mesh Generation Example	Year built:	2023	
Description:	Mesh Generation and Dead Load Only Analysis Training			
Location:	Pittsburgh, PA	Length:	35	ft
Facility carried (7):		Route number:	376	
Feat. intersected (6):		Mi. post:		
Default units:	US Customary V			
Bridge associa	ation BrR 🗸 BrD 🗌 BrM		OK	Apply Cancel

First define the components. This structure has 1 appurtenance, 2 steel shapes, a concrete material, and a steel material.

Appurtenances

Parapet: (Use 'Copy from library...' to save time.)

🗛 Bridge Ap	purtenances - Parapet		- □ >
Name:	Jersey Barrier		
Description:	Standard New Jersey Barrier		
	All dimensions are in inches		
(B	Additional load:	kip/ft Roadway Surface	Parapet unit load: 0.15 kcf Calculated properties Net centroid (from reference line): 7.880 in Total load: 0.505 kip/ft
		Copy from library	OK Apply Cancel

Steel Shapes:

Angles:

$I = 5v = 5v = 1/2 \cdot (2)$	IIce	Conv	from	library ,	to save	time)
LJXJX1/2. (Use	Сору	nom	norary	to save	time.)



I Shapes:

W16x67: (Use 'Copy from library...' to save time.)

Description: W 1	6x67 Imported from AISC Tables (2011) Properties 1 in 1 1.07 0.665 in Y 1.07	W shape M shape S shape HP shape
Dimensions	Properties	in
		in
	0.395 in Y 10.2 in Y 10.2	in

Materials:

Concrete:

🕰 Bridge Ma	terials - Concrete			-	×
Name:	Class A (US)				
Description:	Class A cement concret	e			
Compressive	strength at 28 days (f'c):	4.0000006	ksi		
Initial compr	essive strength (f'ci):		ksi		
Composition	of concrete:	Normal ~			
Density (for	dead loads):	0.15	kcf		
Density (for	modulus of elasticity):	0.145	kcf		
Poisson's rat	io:	0.2			
Coefficient o	f thermal expansion (α):	0.000006	1/F		
Splitting ten	sile strength (fct):		ksi		
LRFD Maxim	um aggregate size:		in		
	Compute				
Std modulus	of elasticity (Ec):	3644.149254	ksi		
LRFD module	us of elasticity (Ec):	3986.548657	ksi		
Std initial mo	odulus of elasticity:		ksi		
LRFD initial r	modulus of elasticity:		ksi		
Std modulus	of rupture:		ksi		
LRFD module	us of rupture:	0.479857	ksi		
Shear factor:		1			

Structural Steel:

Grade 50W: (Use 'Copy from library...' to save time.)

de 50W HTO M270 Grade 50W							
HTO M270 Grade 50W							
s							
n yield strength (Fy):	50.000073	ksi					
n tensile strength (Fu):	70.0000102	ksi					
mal expansion:	0.0000065	1/F					
	0.49	kcf					
ity (E):	29000.004206	ksi					
	s n yield strength (Fy): n tensile strength (Fu): mal expansion: ity (E):	s n yield strength (Fy): 50.0000073 n tensile strength (Fu): 70.0000102 mal expansion: 0.0000065 0.49 ity (E): 29000.004206	s n yield strength (Fy): 50.0000073 ksi n tensile strength (Fu): 70.0000102 ksi mal expansion: 0.0000065 1/F 0.49 kcf ity (E): 29000.004206 ksi	s n yield strength (Fy): 50.0000073 ksi n tensile strength (Fu): 70.0000102 ksi mal expansion: 0.0000065 1/F 0.49 kcf ity (E): 29000.004206 ksi	s n yield strength (Fy): 50.0000073 ksi n tensile strength (Fu): 70.0000102 ksi mal expansion: 0.0000065 1/F 0.49 kcf ity (E): 29000.004206 ksi	s n yield strength (Fy): 50.0000073 ksi n tensile strength (Fu): 70.0000102 ksi mal expansion: 0.0000065 1/F 0.49 kcf ity (E): 29000.004206 ksi	s n yield strength (Fy): 50.000073 ksi n tensile strength (Fu): 70.0000102 ksi mal expansion: 0.0000065 1/F 0.49 kcf ity (E): 29000.004206 ksi

With all the bridge components defined, the bridge workspace tree should now include all the following components.



Diaphragm Definitions

Define the diaphragms. Double click on 'Diaphragm Definitions' in the bridge workspace tree to add a new diaphragm definition. This structure has one diaphragm.

Workspace				\$2							
Bridge Components Components											
A Diaphragm Definitions									_		×
Name: K Frame		Diaphrag	m type	Type 2	\sim]	Number o	of elen	nents in fixed me	mber: 4	\sim
Diaphragm types:	Member	Shape	Se	ection	Section	location	Materia				
	AB L	5x5x1/2 🗸	Vertic	al V	Top Left	~	Grade 50W	~			-
	CD L	5x5x1/2 ~	Vertic	al V	Top Left	\sim	Grade 50W	~			
C → D Type: 1	AE L	5x5x1/2 ~	Vertic	al V	Top Left	~	Grade 50W	~			
C Type: 2											
											w
	Connection	Support type	Y (in)	Me	asured from						
Type: 3	А	Pinned 🗸		Top of W	eb	~					-
	В	Pinned 🗸		Top of W	eb	~					
	С	Pinned 🗸		Bottom o	of Web	~					
I I	D	Pinned 🗸		Bottom o	of Web	\sim					
Type: 4	E	Pinned ~				\sim					
											Ţ
								DK	Apply	Car	ncel

Superstructure Definition

Create a new girder system superstructure definition in the bridge workspace tree.

C	
Workspace	
Bridge Components	
 Mesh Generation Components L Sx5x1/2 L Sx5x1/2 T W 16x67 Class A (US) I Grade 50W Diaphragm Definitions X Frame Lateral Bracing Definitions SUPERSTRUCTURE DEFINITION BRIDGE ALTERNATIVES 	IS
New Superstructure Definition Girder system superstructure Girder line superstructure	X Superstructure definition wizard
Floor system superstructure Floor line superstructure Truss system superstructure Truss line superstructure Reinforced concrete slab system superstructure Concrete multi-cell box superstructure Advanced concrete multi-cell box superstructure	

In the **girder system superstructure definition window**, enter the following information. Make sure to input the horizontal curvature. A girder system structure cannot be modified from straight to curved or curved to straight after it has been defined. Leave the default analysis settings under the 'Analysis' tab. These options will be modified later. Select 'OK' to close and save the window.

		ngine								
lame:	Span 1								Modeling	
									O Multi-girder system MCB	
									With frame structure simplified de	efinit
escription:									Deck type:	
									Concrete Deck 🗸	
efault units:	US Customary	~	Ente	r span	lengths				For PS/PT only	
umber of spans:	1.0		alon	g the r	eference	2			Average humidity:	
umber of airders:	6				Lanath				%	
				Span	(ft)					
			>	1	35	5			Member alt. types	
						_			Steel	
									P/S	
									R/C	
									F/1	
Horizontal curvat	ture along reference	line								
Horizontal curvat	ture along reference curvature	e line —	ance fro	om PC	to first s	support line		ft		
Horizontal curvat Horizontal c Superstructur	ture along reference curvature re alignment	e line Dista Start	ance fro	om PC nt leng	to first s jth:	support line	0	ft ft		
Horizontal curvat Horizontal c Superstructur	ture along reference curvature re alignment	e line Dista Start Radii	ance fro tange	om PC nt leng	to first s pth:	support line	0	ft ft ft		
Horizontal curvat Horizontal c Superstructur C Curved Tangent	ture along reference curvature ire alignment t, curved, tangent	line Dista Start Radii Direo	ance fro : tange us: ction:	om PC nt leng	to first s pth:	support line	0 900 Right ~	ft ft ft		
Horizontal curvat Horizontal c Superstructur Curved Tangent Tangent	ture along reference curvature rre alignment t, curved, tangent t, curved	line Dista Start Radii Direc End t	ance fro : tange us: ction: tangen	om PC nt leng t lengt	to first s jth: th:	upport line	0 900 Right ~ 0	ft ft ft ft		
Horizontal curvat Horizontal c Superstructur Curved Curved	ture along reference curvature ire alignment t, curved, tangent t, curved , tangent	line Dista Start Radii Direc End 1 Dista	ance fro tanget us: ction: tangen ance fro	om PC nt leng t lengt	to first s jth: th:	upport line t line to PT:	0 900 Right ~ 0	ft ft ft ft		
Horizontal curvat Horizontal c Superstructur Curved Tangent Curved,	ture along reference curvature ire alignment t, curved, tangent t, curved t, tangent	line Dista Start Radii Direc End t Dista Desig	ance fro tange us: ction: tangen ance fro gn spe	om PC nt leng t lengt om last ed:	to first s pth: th: t suppor	support line t line to PT:	0 900 Right ~ 0 45	ft ft ft ft ft mph		
Horizontal curvat Horizontal c Superstructur Curved Tangent Curved,	ture along reference curvature re alignment t, curved, tangent t, curved t, curved t, tangent	e line Dista Start Radii Direc End 1 Dista Desig Supe	ance fro : tangen us: ction: tangen ance fro gn spe ereleva	om PC nt lengt om last ed: tion:	to first s gth: th: t suppor	support line t line to PT:	0 900 Right ~ 45 5.8	ft ft ft ft ft %		

3DFEM5 – Mesh Generation and Dead Load Analysis Example

Expand the tree for the new girder system structure definition.



Load Case Description

Add the default load case descriptions for the girder system superstructure.

	Load case name	Description	Stage	Тур	e	Time* (days)	
1	DC1	DC acting on non-composite section	Non-composite (Stage 1) 🛛 🗸 🗸	D,DC	~		
	DC2	DC acting on long-term composite section	Composite (long term) (Stage 2) 🛛 🗸	D,DC	~		
	DW	DW acting on long-term composite section	Composite (long term) (Stage 2) 🛛 🗸	D,DW	~		
,	CID Forms	Weight due to stay in place forms	NI 01/01 43 V				
	SIF FORTIS	weight due to stay-in-place forms	Non-composite (stage 1)	D,DC	×]		
	ar roms	weight due to stay-in-place forms	Non-composite (stage 1)	D,DC	~		

Framing Plan Detail

Enter the framing plan details in the **Structure Framing Plan Details window**. First, input the following information in the 'Layout' tab. Select 'Apply' to save the layout to memory and keep the window open.



Next input the diaphragms for the girder system structure. The diaphragms must be input within each girder bay. With 6 girders, this structure will have 5 girder bays. When inputting distances along girders, especially for curved structures, try to be precise. The system length tolerance and the structure FEM node tolerance will be used to determine when two points are equal. These tolerances provide some flexibility when inputting distances. Generally, though, it is best to use consistent inputs when entering data in different windows.

For example, if a girder is 30.0 feet long and the system length tolerance is 0.01 feet, then a diaphragm at 29.97 feet would not be considered at the end of the girder. The finite element model would include nodes at 29.97 ft and 30.0 ft. It is recommended to place the diaphragm at 30.0 feet to match the input for the girder. Using a smaller tolerance, for example, 0.001, would mean separate nodes could be generated at 29.995 ft and 30.000 ft.

Note: The following screenshots were taken after updating the BrDR preferences for the bridge workspace to display the entered number of decimal positions.



Spacing Support reference numb type		port nber	S dis	itart tance (ft)	Left diaphragm spacing	Right diaphragm spacing	Number of spaces	Left length	Right length	E dist (nd tance (ft)	Load (kip)	Diaphra	ıgm			
		num		Left girder	Right girder	(ft)	(ft)		(tt)	(tt)	Left girder	Right girder					
	Both Girders	\sim	1	\sim	0	0	0	0	1	0	0	0	0	0.25	K Frame	\sim	
	Both Girders	\sim	1	\sim	18.99	10.8	0	0	1	0	0	18.99	10.8	0.19	K Frame	\sim	
	Both Girders	\sim	1	\sim	29.78	21.6	0	0	1	0	0	29.78	21.6	0.19	K Frame	\sim	
:	Both Girders	\sim	1	\sim	34.14	34.207486	0	0	1	0	0	34.14	34.207486	0.25	K Frame	\sim	
:	Both Girders Both Girders	~	1	~	29.78 34.14	21.6 34.207486	0	0	1	0	0	29.78 34.14	21.6 34.207486	0.19	K Frame K Frame		~

Girder Bay 2:

de	er bay: 2		\sim	Сору	bay to	Diaphra	gm I								
	Spacing reference	S	Support	S dis	tart tance (ft)	Left diaphragm	Right diaphragm	Number	Left length	Right length	dis	ind tance (ft)	Load (kip)	Diaphragm	
Both Girders Both Girders Both Girders > Both Girders	type	Ľ	umber	Left girder	Right girder	(ft)	(ft)	or spaces	(ft)	(ft)	Left girder	Right girder	(kip)		
	Both Girders	1	~	0	0	0	0	1	0	0	0	0	0.25	K Frame 🔍 🗸	
	Both Girders	1	\sim	21.6	13.42	0	0	1	0	0	21.6	13.42	0.19	K Frame 🔍	1
	Both Girders	1	\sim	34.207	34.3686	0	0	1	0	0	34.207	34.3686	0.25	K Frame 🔍	

$3DFEM5-Mesh\ Generation\ and\ Dead\ Load\ Analysis\ Example$

Girder Bay 3:

								igm d	Diaphra	bay to	Сору	\sim		er bay: 3	rde
	Diaphragm	Load (kip)	ind tance (ft)	dis	Right length	Left length	Number of spaces	Right diaphragm spacing	Left diaphragm spacing	itart itance (ft)	dis	Support) ie	Spacing referenc	
			Right girder	Left girder	(ft)	(ft)		(ft)	(ft)	Right girder	Left girder			type	Both Both Both
	K Frame 🛛 🗸	0.25	0	0	0	0	1	0	0	0	0	1 ~	\sim	Both Girders	
	K Frame 🗸 🗸	0.19	5.24	13.42	0	0	1	0	0	5.24	13.42	1 ~	\sim	Both Girders	
	K Frame \checkmark	0.19	22.35	30.53	0	0	1	0	0	22.35	30.53	1 ~	\sim	Both Girders	
	K Frame \sim	0.25	34.630481	34.368	0	0	1	0	0	34.630481	34.368	1 ~	\sim	Both Girders	ł
~	K Frame K Frame	0.19	22.35 34.630481	30.53 34.368	0	0	1	0	0	22.35 34.630481	30.53 34.368	1 × 1 ×	~	Both Girders Both Girders	>

Girder Bay 4:

rd	er bay: 4	~	Сору	bay to	Diaphra wizaro	gm I									
	Spacing reference	Support	dis	itart tance (ft)	Left diaphragm spacing	Right diaphragm spacing	Number of spaces	Left length	Right length	E dist (nd ance ft)	Load (kip)	Diaphrag	gm	
	type		Left girder	Right girder	(ft)	(ft)		(11)	(rt)	Left girder	Right girder				
	Both Girders \checkmark	1 ~	0	0	0	0	1	0	0	0	0	0.3	K Frame	~	
	Both Girders \sim	1 ~	22.35	12.65	0	0	1	0	0	22.35	12.65	0.23	K Frame	\sim	
	Both Girders \sim	1 ~	34.63	35.077295	0	0	1	0	0	34.63	35.077295	0.3	K Frame	\sim	
/	Both Girders		34,03	33,011233				U		34.03	33.011293	0.3	K Hame		

3DFEM5 - Mesh Generation and Dead Load Analysis Example

Girder Bay 5:

							d	wizard	bay to	Сору	~		er bay:	rde
Diaphrag	Load (kip)	nd tance (ft)	dis	Right length	Left length	Number of spaces	Right diaphragm spacing	Left diaphragm spacing	itart itance (ft)	dis	Support		Spacing reference	
		Right girder	Left girder	(11)	(ft)		(ft)	(ft)	Right girder	Left girder			type	
K Frame	0.3	0	0	0	0	1	0	0	0	0	1 ×	\sim	Both Girders	
K Frame	0.23	2.95	12.65	0	0	1	0	0	2.95	12.65	1 ~	\sim	Both Girders	
K Frame	0.23	15.3	25	0	0	1	0	0	15.3	25	1 ~	\sim	Both Girders	
K Frame	0.3	35.496115	35.077295	0	0	1	0	0	35.496115	35.077295	1 ~	\sim	Both Girders	,
K Frame K Frame	0.23	15.3 35.496115	25	0	0	1	0	0	15.3 35.496115	25	1 ~ 1 ~	~	Both Girders Both Girders	>

Review the framing plan schematic to verify the framing plan details are correct.



Structure Typical Section

Next, define the structure typical section.

Deck:

erhang		ne	finition	Bight ov	erhang			
k Deck (cont'd) Parapet Median	Railing	G	eneric Si	dewalk	Lane position	Striped lanes	Wearing surface	
verstructure definition reference line is wit	hin	`	the bridg	e deck.				
tance from left edge of deck to	Start 9	ft	End 29	ft				
tance from right edge of deck to perstructure definition reference line:	3	ft	13	ft				
t overhang: 2		ft	2	ft				
mputed right overhang: 3	.75	ft	3.75	ft				

Deck (cont'd):

Distance from left edge superstructure definitio	e of deck to jil n ref. line ji eck i← ckness i	Distance from righ superstructure def Superstructure Reference Line	edge of decl nition ref. line						
overhang	pet Media	an Railing	Generic	Bight ov	erhang Lane position	Striped lanes	Wearing surface		
Deck concrete:	Class A (US)			\sim					
Total deck thickness:	10	in							
Load case:	DC1	~							
Deck crack control parameter:		kip/in							
Sustained modular ratio factor:	3								
Deck exposure factor:									

$3DFEM5-Mesh\ Generation\ and\ Dead\ Load\ Analysis\ Example$

Parapet:

Struc	ture Typical Section								-		
ack	Front										
Deck	Deck (cont'd) Parapet	Median	Railing Ger	neric Sidewalk	Lane pos	ition Stripe	ed lanes W	/earing surface			
	Name	Load case	Measure to	Edge of deck dist. measured from	Distance at start (ft)	Distance at end (ft)	Front face orientation				
>	Jersey Barrier 🔍 🗸	DC1 🗸	Back \checkmark	Left Edge 🗸 🗸	0	0	Right \vee				
	Jersey Barrier 🗸 🗸	DC1 🗸	Back \checkmark	Right Ed 🗸	0	0	Left 🗸 🗸				
							(New Duplicate		Delete	
								ОК Арр	ly	Canc	e

Lane Position:

🕰 Stru	cture Typical S	ection				-		Х
Deck	Travelw Deck (con	A) B Superstructure ay 1 Tr Tr Tr Tr Tr Tr Tr Tr Tr Tr	re Definition Reference Line avelway 2	valk Lane position Strip	ed lanes Wearing surface			
	Travelway number	Distance from left edge of travelway to superstructure definition reference line at start (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at start (B) (ft)	Distance from left edge of travelway to superstructure definition reference line at end (A) (ft)	Distance from right edge of travelway to superstructure definition reference line at end (B) (ft)			
>	1	-27.25	11.25	-27.25	11.25		A	
	LRFD fatigue Lanes ava	ailable to trucks:			New	Dunlicate	Delete	
			Compute		OK	Apply	Cance	:

3DFEM5 – Mesh Generation and Dead Load Analysis Example

Review the structure typical section schematic to verify the typical section inputs.

Bridge Typical Section				- U X
i 🗈 📐 🔍 Q, 🕂 🎛 🗟 🛏 111% 🗸				÷
Mesh Generation Mesh Generation Example - Span 1 11/1/2023				
	42'-0"		>	
-Deck Thickness 10"	38'-6"			
	Travelway 1			
2'-0" 6'-9" 6'-9"	6'-9" 8'-0"	8'-0"	3'-9"	
	·	, 		

Stiffener Definitions

Add stiffener definitions to the structure. This structure has one transverse stiffener definition and one bearing stiffener definition.

Transverse Stiffener:

ame: Plate s	tiffener								
Stiffener type	2			Top gap:				7	
Single					in			_	
O Pair				Width:					
				3.5	in	\mapsto			
Plate Thickness:	0.625	in							
Material:	Grade 50W		\sim	Bottom gap:	in			_	
Welds									
Тор:	None	~							
Web:	None	~							
Bottom:	None	\sim							
					OK	Ap	oly	Canc	el

Bearing Stiffener:

ame: Bearin	g Stiffener								
Plate						in			
Thickness:	0.75	in				in			
Material:	Grade 50W	/		~					
					6	in	$ \longrightarrow $		
Welds	N								
lop:	None		~			in			
Web:	None		~			m	│ <mark> </mark>	_/	
Bottom:	None		\sim			in	** **		

Member Loads

For G1 only, enter an additional member load of 0.2kips/ft.

Workspace			☆ × Sch	ematic			
Bridge Components							
Amesh Generation Components Desey Barrier L 5x5x1/2	A Girder Memb	er Loads				— C	1 × 1
	× ×	+ + +	+ + +	+ + +	X		
- ≥ K Frame - Ø Lateral Bracing Definitions - ⊗ SUPERSTRUCTURE DEFINITIONS - ₩ Span 1	Pedestrian Ioa Uniform	d: Distributed	lb/ft Concentrated	Settlement			
Impact/Dynamic Load Allowance - 武 Load Case Description - 歴 Framing Plan Detail		Load case name	Span	Uniform load (kip/ft)	Description		
 	X DC2	~	All Spans 🗸	0.2	Utilities		-
					N	OK Apply C	2 Cancel

Member Alternatives

Now add member alternatives for each of the six girders. Since this structure is a curved structure, each girder has a slightly different total length. Each girder has to be input separately. View the calculated length of each girder by opening the girder member definition windows.

A Member									_	-		×
Member name:	G1			Link with	h: None	```	~					
Description:												
	Existing	Current	Member altern	ative name	Description							
Number of span	s: <u>1</u>	Span no.	Span length (ft)									
	>	1	34.140884									
								ОК	Apply		Cance	el

When defining a girder member alternative, it can be helpful to record the total length of the girder member. For a girder member alternative, the girder profile, deck profile, haunch profile and lateral supports will all typically have a total length equal to the girder length.

In this structure, all girders are schedule based steel plate girders. The web for each girder has a constant depth of 24" and a width of 0.5". The top and bottom flanges have a transition at 5 feet from each end of the span. Both flanges have a width of 8" everywhere. The flange thickness for the first and last ranges is 0.5" and the thickness for the middle range is 0.75".

Girder 1: (Length = 34.140884 ft) Define the member alternative:

ember alternative: Exterior Girder Description Specs Factors Engine Import Control options Description: Material type: Steel Girder type: Plate Modeling type: Multi Girder System Default units: US Customary v Schedule based Left: Right: in Self load Engine Assigned v Lefault rating method: LFR
Description Specs Factors Engine Import Control options Description: Material type: Steel Girder type: Plate Modeling type: Multi Girder System Default units: US Customary v Girder property input method End bearing locations US Customary v Schedule based Left: in Right: in Self load Default rating method: Load case: Engine Assigned LFR Additional self load: %
Description: Material type: Steel Girder type: Plate Modeling type: Multi Girder System Default units: US Customary V Girder property input method Cross-section based Left: in Right: in Self load Load case: Engine Assigned V Additional self load: kip/ft Additional self load: %
Girder type: Plate Modeling type: Multi Girder System Default units: US Customary ~
Modeling type: Multi Girder System Default units: US Customary ~ Girder property input method Cross-section based Left: in Right: in Self load Load case: Engine Assigned Default rating method: LFR Default rating method:
Default units: US Customary Order property input method End bearing locations Left: in Right: Default rating method: Load case: Default rating method: Default rating method:
Girder property input method End bearing locations Schedule based Left:in Cross-section based Right:in Self load Default rating method: Load case: Engine Assigned Additional self load: %
Schedule based Cross-section based Right: In Right: Default rating method: Load case: Engine Assigned Default rating method: Load case: Additional self load: %
Cross-section based Right: in Self load Default rating method: Load case: Engine Assigned V Additional self load: kip/ft Additional self load: %
Self load Default rating method: Load case: Engine Assigned Additional self load: kip/ft Additional self load: %
Self load Default rating method: Load case: Engine Assigned V LFR V Additional self load: kip/ft Additional self load: %
Additional self load: kip/ft Additional self load: %
Additional self load:
Additional self load: /o

Define the girder profile ranges.

	Plate Gird	ler										
/eb	Top fl	ange Bo	ottom flange									
	Begin depth (in)	Depth var	End ry depth (in)	Thickness (in)	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Material	Weld at right		
>	24	None	24	0.5	1 ~	0	34.140884	34.140884	Grade 50W	None 🚿	~	
										New	Duplicate	e Delete
											ОК Арр	ly Cance
irde ≃: /eb	er Profile Plate Gird	ler ange Bo	ottom flange	2							ОК Арр	ly Cance
irde e: /eb	er Profile Plate Gird Top fl. Begin width (in)	ler ange Bo End width (in)	ottom flange Thickness (in)	Support number	Start distance (ft)	Ler (†	ngth ft)	End distance (ft)	Material	Weld	OK App Weld at right	ly Cance
irde e:	er Profile Plate Gird Top fla Begin width (in) 8	ler ange Ba Width (in) 8	ottom flange Thickness (in) 0.5	Support number 1 ~	Start distance (ft)	Ler (1	ngth ft) 5	End distance (ft) 5 (Material Grade 50W V	Weld	Weld at right None V	ly Cance
irde e:	er Profile Plate Gird Top fl Begin width (in) 8 8	er Bange End width (in) 8 8	Thickness (in) 0.5 0.75	Support number 1 ~ 2 1 ~ 2	Start distance (ft)	Ler (t 0 5 24	ngth ft) 5 4.140884	End distance (ft) 5 0 29.140884 0	Material Grade 50W V	Weld None >>	Weld at right None ~	ly Cance
iirde e: Veb	er Profile Plate Gird Top fl Begin width (in) 8 8 8 8	End width (in) 8 8 8 8 8	Thickness (in) 0.5 0.75 0.5	Support number 1 ~ ~ 1 1 ~ 1 1 ~ 1	Start distance (ft) 29.14088	Ler (1) 5 24 34	ngth ft) 5 4.140884 5	End distance (ft) 5 0 29.140884 0 34.140884 0	Material Grade 50W V Grade 50W V Grade 50W V	Weld None > None > None >	Weld at right None ~ None ~	ly Cance
irde e: /eb	er Profile Plate Gird Top fl Begin width (in) 8 8 8 8 8	ler Bange Ba width (in) 8 8 8 8 8	Thickness (in) 0.5 0.75 0.5	Support number 1 ~~ 1 1 ~~ 1 1 ~~ 1	Start distance (ft) 29.14088	0 5 24 34	ngth ft) 5 4.140884 5	End distance (ft) 5 0 29.140884 0 34.140884 0	Material Grade 50W V Grade 50W V Grade 50W V	Weld None \v None \v None \v	Weld at right None ~ None ~	ly Cance
e:	er Profile Plate Gird Top fl Begin width (in) 8 8 8 8 8	End width (in) 8 8 8 8 8	Thickness (in) 0.5 0.75 0.5	Support number 1 ~ 2 1 ~ 2 1 ~ 2 1 ~ 2	Start distance (ft) 29.14088	Ler (1 5 24 34	ngth ft) 5 4.140884 5	End distance (ft) 5 0 29.140884 0 34.140884 0	Material Grade 50W V Grade 50W V	Weld None > None > None >	Weld at right None ~ None ~	e Delete

Tip: The top and bottom flanges are the same. Use the 'Copy to bottom flange' button to copy the ranges from the top flange to the bottom flange.

4	Gird	er Profile											-		×
Т	ype: Web	Plate Gird	ler ange E	ottom flange	2										
		Begin width (in)	End width (in)	Thickness (in)	Sunu	ipport imber	Start distance (ft)	Length (ft)	End distance (ft)	Material	Weld	Weld at right			
	>	8	8	0.5	1	\sim	0	5	5	Grade 50W \sim	None 🗸 🗸	None 🗸			
		8	8	0.75	1	\sim	5	24.140884	29.140884	Grade 50W 🗸 🗸	None 🗸	None 🗸			
		8	8	0.5	1	\sim	29.140884	5	34.140884	Grade 50W 🗸 🗸	None 🗸 🗸	None 🗸			
		Copy to	top flang	e							New	Duplicate	<u>+</u>	Delete	
												ОК Арр	ly	Cance	9

$3DFEM5-Mesh\ Generation\ and\ Dead\ Load\ Analysis\ Example$

Deck Profile

Enter the deck profile.

concrete															
	Reint	orcement	Shear co	nnectors											
Materia	al	Support number	Start distance (ft)	Length (ft)	End distance (ft)	Structural thickness (in)	Start effective flange width (Std) (in)	End effective flange width (Std) (in)	Start effective flange width (LRFD) (in)	End effective flange width (LRFD) (in)	n				
Class A (US)	\sim	1 ~	0	34.140884	34.140884	9.5	64.5	64.5	64.5	64.5					
ompute fror	n										New	Duplicat	e	Delete	
	Materia Class A (US)	Material Class A (US) 🗸 🗸	Material Support Llass A (US) × 1 ×	Material Support number Start distance (ft) Class A (US) 1 0	MaterialSupport numberStart distance (ft)Length (ft)Zlass A (US) ∨1 ∨034.140884	MaterialSupport numberStart distance (ft)Length (ft)End distance (ft)Class A (US) ∨1 ∨034.14088434.140884	Material Support number Start distance (ft) Length (ft) End distance (ft) Structural thickness (in) Class A (US) V 1 V 0 34.140884 34.140884 9.5	MaterialSupport numberStart distance (ft)Length (ft)End distance (ft)Structural thicknessStructural fange width (std) (in)Class A (US) ∨1 ∨034.14088434.1408849.564.5	MaterialSupport numberStart distance (ft)Length (ft)End distance (ft)Structural distance (ft)Start effective flange width (Std) (in)End effective flange width (Std) (in)Class A (US) ∨1 ∨034.14088434.1408849.564.5	MaterialSupport numberStart distance (ft)Length (ft)End distance (ft)Structural thicknessStart effective flange width (Std) (in)Start effective flange width (IRFD) (in)Class A (US) v1 v034.14088434.1408849.564.564.564.5	MaterialSupport numberStart distance (ft)Length (ft)End distance (ft)Structural distance (ft)Structural thickness (in)Struct effective flange width (Std) (in)Start effective flange width (LRFD) (in)End effective flange width (LRFD) (in)Class A (US) ∨1 ∨034.1408849.564.564.564.564.5	MaterialSupport numberStart distance (ft)Length distance (ft)End distance (ft)Start effective ftange width (in)End effective ftange width (Std) (in)End effective ftange width (IRFD) (in)Ind effective ftange width (IRFD) (in)Class A (US) v1 v034.1408849.564.564.564.564.5	MaterialSupport numberStart distance (ft)Length distance (ft)End distance (ft)Structural thicknessStart effective flange width (Std) (jn)End effective flange width (LRFD) (jn)End effective flange width (jn)End effective flange width (jn)Zlass A (US) v1 v034.1408849.564.564.564.5<	MaterialSupport numberStart distance (ft)Length distance (ft)End distance (ft)Start effective flange width (Std) (in)End effective flange width (ILRFD) (in)In effective flange width (ILRFD) (in)Class A (US) v1 v034.1408849.564.564.564.564.51	MaterialSupport numberStart distance (ft)Length distance (ft)End distance (ft)Structural thickness (in)Start effective flange width (Std) (in)End effective flange width (LRFD) (in)End effective flange width (in)End effective flange width (in)End effective flange widt

Define the composite region.

e:	Plate											
eck	concrete	e Reinforc	ement Shear conne	ectors								
	Support	t distance (ft)	Length (ft)	End distance (ft)	Connector ID	Number of spaces	Number per row	Transverse spacing (in)				
>	1 ~	0	34.140884	34.140884	Composite 🗸							-
									New	Duplicate D	Delete	

Stiffener Ranges

Add the stiffeners. Use the 'Apply at diaphragms...' button in the **Stiffener Ranges window** to add the transverse stiffeners at diaphragm locations and the bearing stiffeners at support locations.

	ier Ranges									-	
s s	tart Distance	Sp.	acing								
Transv	erse stiffener ra	anges	Longitu	ıdinal stiffer	ner ranges						
	Name		Support number	Start distance (ft)	Number of spaces	Spacing (in)	Length (ft)	End distance (ft)			
F	Plate stiffener	\sim	1 ~	18.99	1	0	0	18.99			^
> F	Plate stiffener	\sim	1 ~	29.78	1	0	0	29.78			
Ap	oply at S	tiffene	ers between								•

Girders 2-6:

Follow the same steps as shown for girder 1 to define member alternatives for girders 2 through 6. Refer to the tables below for the girder and deck dimensions. All members should have a composite deck along their full length and have stiffeners at diaphragm locations.

Verify the length of each girder matches the length shown in the table below. If the length is different, review the inputs for the Girder System Superstructure Definition window and the Structure Framing Plan Details window.

Table: Computed Girder Lengths

Girder	Length
	(ft)
G1	34.140884
G2	34.207486
G3	34.3686
G4	34.630481
G5	35.077295
G6	35.496115

Table: Web Profile Definitions

Girder	Range Start	Range Length	Depth	Thickness
	(ft)	(ft)	(in)	(in)
G1	0.0	34.140884	24.0	0.5
G2	0.0	34.207486	24.0	0.5
G3	0.0	34.3686	24.0	0.5
G4	0.0	34.630481	24.0	0.5
G5	0.0	35.077295	24.0	0.5
G6	0.0	35.496115	24.0	0.5

Girder	Range Start	Range Length	Width	Thickness
	(ft)	(ft)	(in)	(in)
	0.0	5.0	8.0	0.5
G1	5.0	24.140884	8.0	0.75
	29.140884	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G2	5.0	24.207486	8.0	0.75
	29.207486	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G3	5.0	24.3686	8.0	0.75
	29.3686	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G4	5.0	24.630481	8.0	0.75
	29.630481	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G5	5.0	25.077295	8.0	0.75
	30.077295	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G6	5.0	25.496115	8.0	0.75
	30.496115	5.0	8.0	0.5

Table: Top and Bottom Flange Profile Definitions

Table: Deck Profile Effective Width

			Star	ndard	LR	FD
Girder	Range	Range	Start Effective	End Effective	Start Effective	End Effective
	Start	Length	Flange Width	Flange Width	Flange Width	Flange Width
	(ft)	(ft)	(in)	(in)	(in)	(in)
G1	0.0	34.140884	64.5	64.5	64.5	64.5
G2	0.0	34.207486	81.0	81.0	81.0	81.0
G3	0.0	34.3686	81.0	81.0	81.0	81.0
G4	0.0	34.630481	88.5	88.5	88.5	88.5
G5	0.0	35.077295	96.0	96.0	96.0	96.0
G6	0.0	35.496115	93.0	93.0	93.0	93.0

This completes the data entry for the curved girder system structure. Now would be a good time to save the bridge to the database.



Mesh Generation and Dead Load Only Analysis

Return to the 'Analysis' tab of the Girder System Superstructure Definition window.

Characteristic allele Abiologica	Number of shall alcounts
Consider structural slab thickness for rating	In the deck between girders
Consider structural slab thickness for design	Slower Facter
Wearing surface	More accurate Less accurate
Consider wearing surface for rating	
Consider wearing surface for design	
Consider striped lanes for rating	Target aspect ratio for shell elements
Default analysis type: Line Girder	Slower Faster
	More accurate
Vahisla incrementi 1 ft	1 1.5 2 2.5 3 3.5 4
Transverse loading	3D FE node generation tolerance
Vehicle increment in lane: 2 ft	Vercentage
Lane increment:	
	Span Length Tolerance
3D analysis control options	
LFR: Model non-composite regions as non-composite	7 1 33 0.1
LRFD: Model non-composite regions as non-composite	
LRFR: Model non-composite regions as non-composite	
	3D bracing member end connection analysis
	Calculated factored member force effects
	Maximum of average (stress + strength) and 75% resistance
	Bracing member LRFR factors
	Condition factor: Good or Satisfactory \sim
	Field measured section properties

Use the 'F1' key to open the window help. The help window includes diagrams and descriptions to describe how these options are used in the analysis. Review the help for the different analysis settings shown in this window.

Modify the deck shell options to generate the coarsest mesh. This may not yield the most accurate results. The results from the coarse mesh can become a benchmark by which to compare finer meshes. Since the analysis will run fastest with the coarse mesh, this will also serve as a quick test to verify the model inputs before attempting a larger analysis.

Numbe	er of sh	ell ele	ments						
O In	the de	eck be	tween g	girders					
() In	the w	eb bet	ween f	langes					
Slower							F	aster	
More a	ccurat	e					L	ess ac	curate
10	9	8	7	6	5	4	3	2	-••
Target	aspect	ratio f	or shel	l eleme	ents –				
Slower							F	aster	
Slower More a	iccurat	e					F	aster .ess ac	curate
Slower More a	accurat	e					F	aster .ess ac	curate

Save the coarse shell mesh generation options to memory by clicking 'OK' on the window.

The dead load only analysis can be used to quickly compare the accuracy of finite element models with increasing levels of mesh refinement. Once an appropriate mesh is selected for the dead load analysis, the live load analysis can be run with greater confidence in the results.

Several techniques can be used to find acceptable options for the mesh generation.

- Inspect the finite element model, checking for appropriate aspect ratios and minimal element distortion.
- Verify the model geometry accurately represents the structure geometry.
- Compare dead load results. Select several dead load cases. Review the results plots for different members to verify the computed actions are reasonable.
- Copy the results from the tabular results and compare the dead load at different levels of mesh refinement. If the differences in dead load between trials is sufficiently small, further mesh refinement may not be necessary.

Open the analysis settings window and select a 3D dead load only analysis. For a dead load only analysis, vehicles do not need to be added to the Vehicle summary.

Analysis Settings	>
Design review O Rating	Rating method: LFR ~
Analysis type: 3D FEM Lane / Impact loading type: As Requested	Analysis option: DL Only v Apply preference setting: None v
Vehicles Output Engine Description	
Traffic direction: Both directions	Refresh Temporary vehicles Advanced
 Image: Standard Alternate Military Loading EV2 EV3 H 15-44 H 20-44 HS 20-44 HS 20-44 NRL SU4 SU5 SU6 SU7 Type 3.3 Type 3.3 Type 3.32 Agency User defined Temporary 	Add to Add to Add to Remove from
Reset Clear Open template Save	template OK Apply Cancel

Analyze the girder system superstructure.

BRIDGE WORKSPACE	WORKSPACE TO	IOLS VIEW	ANALYSIS DESIGN/RATE	REPORTS REPORTING
Analysis Settings Analysis Analysis	s Tabular Specificati Results Check De	on Engine Res ail Outputs Gr Results	sults Save aph Results	
Workspace				\$ ×
iii ∰ Diaphragm D → 1 Unpact iii hr Spani → SupersTRUC iii hr Spani → Load C → Branin → Branin → Branin → Branin → Supers → Supers → Load C → Branin → Branin → Branin → Supers → MEMB iii hr Supers → Supers	finitions p Definitions VDynamic Load Allows are Description g Plan Detail) Deterioration) Spec Check Selection re Typical Section tructure Loads Connector Definitions ERS Member Loads	ince		

After the analysis is complete, open the Engine Outputs window.



The S2 Span 3D FE Model Graphics opens a viewer which displays the stage 2 composite finite element model, and the Model Generation Node Merge Report opens a text report on the mesh generation for the girders. Open both and review the output.

∎ s	pan 1
	AASHTO_LFR_3D
	Model Generation Node Merge Report
	-S1 Span 3D Model
	S1 Span 3D FE Model Graphics
	S1 Span 3D Model Actions
	S2 Span 3D Model
	S2 Span 3D FE Model Graphics
	S2 Span 3D Model Actions
	Log File
	₽-G1
	IExterior Girder
	8-62

Node Merge Report

The node merge report has several sections. The summary section indicates if all nodes in the final mesh are within the desired merge tolerance. When using a larger tolerance to merge nodes, the program may have to exceed the tolerance to preserve section change or support locations. As a result, some points such as diaphragms or tenth points could be shifted a slightly larger distance to maintain equal nodes on each girder. If this occurs the summary section will highlight these points.

The member nodes section reports the process of adding points for each girder member. Points that are within the system length tolerance are merged and considered at the same location.

ModelGenNodeMe	ergeReport - Notepad				
File Edit Format	View Help				
Model Generatio	n Node Merge Reg	oort			
Summary					
SUCCESS: All gi	rder nodes are d	defined within	the desired t	toleranc	e!
Ŭ					
Member Nodes					
System Length To	olerance = 0.001	L000 (ft)			
G1					
Distance	Source Type	Node Type	Added At	Span	Member Node
(ft)			(ft)		In Span
0.000000	SUPPORT	NEW	0.000000	1	1
0.000000	DIAPHRAGM	MERGE	0.000000	1	
0.000000	SECTION CHANGE	MERGE	0.000000	1	
0.000000	TENTH	MERGE	0.000000	1	
3.414088	IENIH	NEW	3.414088	1	2
5.000000	SECTION CHANGE	NEW	5.000000	1	3
6.8281//	TENTH	NEW	6.8281//	1	4
10.242265	TENTH	NEW	10.242265	1	5
13.656354	TENTH	NEW	13.656354	1	6
17.070442	TENTH	NEW	17.070442	1	7
18.990000	DIAPHRAGM	NEW	18.990000	1	8
20.484530	TENTH	NEW	20.484530	1	9
23.898619	TENTH	NEW	23.898619	1	10
27.312707	TENTH	NEW	27.312707	1	11
29.140884	SECTION CHANGE	NEW	29.140884	1	12
29.780000	DIAPHRAGM	NEW	29.780000	1	13
30.726796	TENTH	NEW	30.726796	1	14
34.140884	SUPPORT	NEW	34.140884	1	15
34.140884	IENIH	MERGE	34.140884	1	
34.140884	SECTION CHANGE	MERGE	34.140884	1	
34.140000	DIAPHRAGM	MEKGE	34.140884	1	

The final section of the report lists the nodes along each girder after merging nodes to be at equal percentages along each span of each girder in the structure. When the source type for a node is "Added for equal percent", a node was added to the girder to match the percent for a required node along a different girder.

Mc Mc	odelGenNodeMergeRe	eport - Notepad								- 0	×
File E	dit Format View	Help									
Struc	ture Nodes										Â
Span	Length (ft)	Percent Tolerance (%)	Equivalent	: Length Tole (ft)	erance						
1	35.000000	0.100000		0.035000							
G1 											
Node	Member Node	s Distance (ft)	X (ft)	Y (ft)	Z (ft)	Span	Percent (%)	Max Shift (ft)	Source Type		
1	1	0.000000	-32.753619	0.000000	-27.000000	1	0.000000	0.000000	SECTION CHANGE, SUPPORT,	TENTH,	DIAPI
2	n/a	2.837370	-29.916249	0.000000	-27.000000	1	8.310769	n/a	ADDED FOR EQUAL PERCENT		
3	2	3.414088	-29.339531	0.000000	-27.000000	1	10.000000	0.000000	TENTH		
4	n/a	4.809102	-27.944518	0.000000	-27.000000	1	14.086049	n/a	ADDED FOR EQUAL PERCENT		
5	n/a	4.866522	-27.887097	0.000000	-27.000000	1	14.254235	n/a	ADDED FOR EQUAL PERCENT		
6	n/a	4.929311	-27.824308	0.000000	-27.000000	1	14.438148	n/a	ADDED FOR EQUAL PERCENT		
7	3	5.000000	-27.753619	0.000000	-27.000000	1	14.645198	0.000000	SECTION CHANGE		
8	n/a	5.165918	-27.587701	0.000000	-27.000000	1	15.131179	n/a	ADDED FOR EQUAL PERCENT		
9	4	6.828177	-25.925442	0.000000	-27.000000	1	20.000000	0.000000	TENTH		
10	5	10.242265	-22.511354	0.000000	-27.000000	1	30.000000	0.000000	TENTH		
11	n/a	10.778972	-21.974647	0.000000	-27.000000	1	31.572036	n/a	ADDED FOR EQUAL PERCENT		
12	n/a	12.312300	-20.441319	0.000000	-27.000000	1	36.063214	n/a	ADDED FOR EQUAL PERCENT		
13	n/a	13.331083	-19.422536	0.000000	-27.000000	1	39.047270	n/a	ADDED FOR EQUAL PERCENT		
14	6	13.656354	-19.097266	0.000000	-27.000000	1	40.000000	0.000000	TENTH		
15	n/a	14.715851	-18.037768	0.000000	-27.000000	1	43.103309	n/a	ADDED FOR EQUAL PERCENT		
16	7	17.070442	-15.683177	0.000000	-27.000000	1	50.000000	0.000000	TENTH		
17	8	18,990000	-13.763619	0.000000	-27.000000	1	55.622461	0.000000	DIAPHRAGM		
18	9	20.484530	-12.269089	0.000000	-27.000000	1	60.000000	0.000000	TENTH		
19	n/a	21.557945	-11.195675	0.000000	-27.000000	1	63.144073	n/a	ADDED FOR EQUAL PERCENT		
20	n/a	22.034021	-10.719598	0.000000	-27.000000	1	64.538519	n/a	ADDED FOR EQUAL PERCENT		
21	10	23.898619	-8.855000	0.000000	-27.000000	1	70.000000	0.000000	TENTH		

3D Model Graphics

Use the toolbar in the Model Viewer to scale, pan, and rotate the finite element model.





To better view the meshing of the deck shell elements, use the 'Show Top' option in the ribbon.

From this view we can see the deck is composed of a series of closely spaced elements. Adding additional elements in the deck between the girders would improve the mesh.

The finite element model for this mesh has 1156 total nodes.

Results Graph

To review the results plot, select a member alternative in the bridge workspace tree and open the **Results Graph** from the Design/Rate ribbon.



This is the moment from the additional member uniform load assigned to G1 for the utilities.



Tabular Results

The same load case data is available in a tabular format in the Tabular Results window.



🕰 Analysis Results - Exterior Girder \times Print Print Report type: Stage Dead Load Case Dead Load Actions Load Case 1 - Member Dist'd L 🗸 🗸 Composite (long term) (Stage 2 🗸 Location % Moment Shear Axial Torsior Reaction X Deflection Y Deflection Side Span Span (ft) (kip-ft) (kip) (kip) (kip-ft) (kip) (in) (in) 0.00 0.0 Right -6.40 2.58 2.57 0.00 2.86 0.0000 0.0000 -0.0043 3.41 10.0 2.08 0.0000 Left 2.17 -2.20 0.00 3.41 10.0 Right 2.17 1.89 2.20 0.00 0.0000 -0.0043 5.00 14.6 Left 4.66 1.82 -2.16 0.00 0.0000 -0.0061 5.00 14.6 4.66 1.80 2.16 0.00 0.0000 -0.0061 Right 7.18 1.77 0.0000 -0.0079 6.83 20.0 -2.10 0.00 Left 7.18 0.00 0.0000 -0.0079 6.83 20.0 Right 1.26 2.10 10.24 30.0 Left 11.13 1.42 -1.92 0.00 0.0000 -0.0107 1 10.24 30.0 Right 11.13 1.03 1.92 0.00 0.0000 -0.0107 13.66 40.0 Left 13.33 0.70 -1.68 0.00 0.0000 -0.0125 40.0 Right 13.66 13.33 0.56 1.68 0.00 0.0000 -0.0125 1 17.07 50.0 Left 14.41 0.41 -1.42 0.00 0.0000 -0.0131 50.0 Right 17.07 14.41 -0.02 1.42 0.00 0.0000 -0.0131 18.99 55.6 14.37 0.08 -1.32 0.00 0.0000 -0.0129 Left 18.99 55.6 Right 14.37 -0.25 1.32 0.00 0.0000 -0.0129 20.48 60.0 13.98 -0.18 -1.27 0.00 0.0000 -0.0124 Left 20.48 13.98 -0.44 1.27 0.0000 -0.0124 60.0 Right 0.00 -0.65 -1.22 0.0000 23.90 70.0 11.98 0.00 -0.0106 Left 0.0000 23.90 70.0 Right 11.98 -0.88 1.22 0.00 -0.0106 27.31 80.0 Left 8.53 -1.18 -1.32 0.00 0.0000 -0.0077 1 0.0000 27.31 80.0 Right 8.53 -1.65 1.32 0.00 -0.0077 1 1 29.14 85.4 Left 6.06 -1.72 -1.41 0.00 0.0000 -0.0059 AASHTO LFR 3D Engine Version 7.5.0.3001 Analysis preference setting: None Close

3DFEM5 - Mesh Generation and Dead Load Analysis Example

This table can easily be copied into another program to compare the results for different levels of mesh refinement. To select the entire grid, click the top leftmost cell in the table header. To select a portion of the grid, click and drag over the desired cells. Copy the selected cells using the Ctrl-C on your keyboard and paste into another program using Ctrl-V.

NOTE:

Even though this is a single span structure, the 3D analysis may compute non-zero moments at the supports. These moments at the supports are the result of the structure geometry and the modeled support conditions. A pinned 3D support is assumed to constrain the displacement of the bottom flange of the girder. This support is slightly different from a line girder analysis where a girder is supported at the neutral axis and the bottom flange is free to lengthen under positive flexure.

In the following steps we compare the computed moment at location 0.00 ft as an example of one method of validating the mesh refinement. This location was selected as an example only because it is at the top of the table, any location or action type could be considered.

Refine the finite element mesh by doubling the number of elements in the deck between the girders.

– Number	of she	ll ele	ments						
O In t	the de	k be	tween g	irder	5				
	the we	b bet	ween fl	anges					
Slower More ad	curate						F	aster less acc	curate
10	9	8	7	6	5	4	3	2	' <u>1</u>
Target a	spect r	atio f	for shell	elem	ents –				
Slower							F	aster	
More ad	curate						l	less acc	curate
1	1.	5	2		2.5	3		3.5	4

Save the analysis settings and re-analyze the structure with the dead load only analysis.



The model is improved with double the number of elements between girders. Comparing the moment for the applied uniform dead load from utilities at the first support, the computed moment is about 11.5% larger than in the first case. With such a large difference in moment, consider further mesh refinement.

This finite element model has 1311 total nodes, about 13% more than the coarse finite element model.

🕰 Analysis Results - Exterior Girder _ \times Print Print Report type: Stage Dead Load Case \sim Dead Load Actions Composite (long term) (Stage 2 🗸 % Moment Shear Axial Torsion Reaction X Deflection Y Deflection Location Span Side Span (ft) (kip-ft) (kip) (kip) (kip-ft) (kip) (in) (in) -7.14 2.66 0.0 Right 2.48 0.00 0.0000 0.0000 1 0.00 2.95 2.01 3.41 10.0 2.37 -2.44 0.00 0.0000 -0.0045 1 Left 1 3.41 10.0 Right 1.72 2.15 2.49 0.00 0.0000 -0.0045 1 5.00 14.6 Left 4.73 1.99 -2.52 0.00 0.0000 -0.0064 14.6 Right 0.0000 -0.0064 5.00 4.76 1.96 2.51 0.00 1 1 6.83 20.0 Left 7.61 1.78 -2.54 0.00 0.0000 -0.0083 1 6.83 20.0 Right 7.44 1.39 2.47 0.00 0.0000 -0.0083 -2.47 -0.0113 1 10.24 30.0 Left 11.51 1.39 0.00 0.0000 30.0 Right 1.11 2.38 0.0000 -0.0113 1 10.24 11.59 0.00 1 13.66 40.0 Left 14.01 0.69 -2.22 0.00 0.0000 -0.0133 13.66 40.0 Right 14.07 0.59 2.16 0.00 0.0000 -0.0133 1 17.07 50.0 14.93 0.35 -2.02 0.00 0.0000 -0.0139 1 Left 1 17.07 50.0 Right 15.21 0.04 1.90 0.00 0.0000 -0.0139 1 18.99 55.6 Left 14.88 0.04 -1.90 0.00 0.0000 -0.0137 0.0000 18.99 15.15 -0.24 1.89 0.00 -0.0137 1 55.6 Right 1 20.48 60.0 Left 14.51 -0.24 -1.89 0.00 0.0000 -0.0132 1 20.48 60.0 Right 14.76 -0.46 1.85 0.00 0.0000 -0.0132 0.0000 -0.0113 1 23.90 70.0 Left 12.50 -0.80 -1.82 0.00 12.79 1.82 0.00 0.0000 -0.0113 23.90 70.0 Right -1.01 1 27.31 80.0 Left 8.49 -1.37 -1.87 0.00 0.0000 -0.0082 80.0 Right -1.99 2.00 0.00 0.0000 -0.0082 27.31 9.29 1 -2.01 -2.00 0.00 0.0000 -0.0063 1 29.14 85.4 Left 5.68 AASHTO LFR 3D Engine Version 7.5.0.3001 Analysis preference setting: None Close

3DFEM5 - Mesh Generation and Dead Load Analysis Example

Try doubling the number of elements between girders again, from 2 elements to 4 elements.



rint				<i>c</i> .					1.0			
rt type	21 1. A			Stage		1.00	-	Dead	load Case	D: 111		
Load	Actions		~	Composite	long te	erm) (St	age 2 🗸	Load	Case I - Memb	er Dist'd Li 🗸 🗸		
Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)		
1	0.00	0.0	Right	-7.32	2.60	3.01	0.00	2.98	0.0000	0.0000		
1	3.41	10.0	Left	1.08	2.27	-2.91	0.00		0.0000	-0.0046		
1	3.41	10.0	Right	1.61	2.07	2.83	0.00		0.0000	-0.0046		
1	5.00	14.6	Left	4.65	1.93	-2.77	0.00		0.0000	-0.0065		
1	5.00	14.6	Right	4.63	1.91	2.77	0.00		0.0000	-0.0065		
1	6.83	20.0	Left	7.60	1.75	-2.73	0.00		0.0000	-0.0085		
1	6.83	20.0	Right	7.51	1.37	2.63	0.00		0.0000	-0.0085		
1	10.24	30.0	Left	11.62	1.37	-2.63	0.00		0.0000	-0.0116		
1	10.24	30.0	Right	11.69	1.10	2.51	0.00		0.0000	-0.0116		
1	13.66	40.0	Left	14.18	0.70	-2.30	0.00		0.0000	-0.0135		
1	13.66	40.0	Right	14.23	0.60	2.23	0.00		0.0000	-0.0135		
1	17.07	50.0	Left	15.12	0.37	-2.09	0.00		0.0000	-0.0142		
1	17.07	50.0	Right	15.38	0.05	1.97	0.00		0.0000	-0.0142		
1	18.99	55.6	Left	15.08	0.05	-1.97	0.00		0.0000	-0.0140		
1	18.99	55.6	Right	15.34	-0.20	1.88	0.00		0.0000	-0.0140		
1	20.48	60.0	Left	14.76	-0.20	-1.88	0.00		0.0000	-0.0135		
1	20.48	60.0	Right	14.97	-0.40	1.84	0.00		0.0000	-0.0135		
1	23.90	70.0	Left	12.82	-0.74	-1.80	0.00		0.0000	-0.0115		
1	23.90	70.0	Right	13.05	-0.96	1.79	0.00		0.0000	-0.0115		
1	27.31	80.0	Left	8.87	-1.29	-1.80	0.00		0.0000	-0.0084		
1	27.31	80.0	Right	9.39	-1.82	1.87	0.00		0.0000	-0.0084		
1	29.14	85.4	Left	6.14	-1.84	-1.87	0.00		0.0000	-0.0064		

Comparing the same load case, the moment at the first support increased by about 2.5% from the previous trial.

This finite element model has 1621 total nodes, almost 24% more than the previous finite element model.

Repeat this process and double the number of elements between the girders from 4 to 8.



This finite element model has 2241 total nodes, or 38% more than the model with 4 elements between girders.

🕰 Analysis Results - Exterior Girder _ \times Print Print Report type: Stage Dead Load Case Dead Load Actions Composite (long term) (Stage 2 🗸 🗸 \sim % Location Moment Shear Axial Torsion Reaction X Deflection Y Deflection Span Side (ft) Span (kip-ft) (kip) (kip) (kip-ft) (kip) (in) (in) -7.34 1 0.00 0.0 Right 2.60 3.04 0.00 2.98 0.0000 0.0000 1.63 3.41 2.25 -2.98 0.00 0.0000 -0.0046 1 10.0 Left 0.0000 1 3.41 10.0 Right 1.58 2.07 2.95 0.00 -0.0046 1 5.00 14.6 Left 4.63 1.92 -2.87 0.00 0.0000 -0.0065 14.6 Right 4.62 1.90 0.0000 1 5.00 2.86 0.00 -0.0065 6.83 7.62 1.75 -2.79 0.00 0.0000 20.0 -0.0085 1 Left 6.83 7.52 1.38 2.65 0.00 0.0000 -0.0085 1 20.0 Right 1 10.24 30.0 Left 11.61 1.38 -2.65 0.00 0.0000 -0.0116 1 10.24 30.0 Right 11.69 1.11 2.52 0.00 0.0000 -0.0116 13.66 40.0 14.20 0.70 -2.28 0.00 0.0000 -0.0136 1 Left 13.66 40.0 Right 14.25 0.61 2.24 0.00 0.0000 -0.0136 1 1 17.07 50.0 Left 15.13 0.37 -2.11 0.00 0.0000 -0.0142 17.07 0.06 1.97 0.00 0.0000 -0.0142 1 50.0 Right 15.39 -1.97 0.0000 -0.0140 1 18.99 55.6 Left 15.10 0.06 0.00 1 18.99 55.6 Right 15.36 -0.20 1.88 0.00 0.0000 -0.0140 0.0000 1 20,48 60.0 Left 14.78 -0.20 -1.88 0.00 -0.0136 1 20.48 60.0 Right 14.99 -0.40 1.83 0.00 0.0000 -0.0136 23.90 70.0 Left 12.85 -0.74 -1.79 0.00 0.0000 -0.0116 1 1 23.90 70.0 Right 13.08 -0.96 1.79 0.00 0.0000 -0.0116 27.31 80.0 8.85 -1.29 -1.79 0.00 0.0000 -0.0085 1 Left 27.31 80.0 Right 9.38 -1.82 1.81 0.00 0.0000 -0.0085 1 29.14 85.4 6.18 -1.83 -1.81 0.00 0.0000 -0.0065 1 Left AASHTO LFR 3D Engine Version 7.5.0.3001 Analysis preference setting: None Close

3DFEM5 – Mesh Generation and Dead Load Analysis Example

In this case there was only a 0.27% change in moment. At this point, further refinement may not be necessary. Using 4 elements and 8 elements yields approximately the same result, but 4 elements will analyze faster than the 8-element mesh.

The target aspect ratio is the next shell meshing option. It is always measured in the longitudinal direction.



Because this model has a short span length and a lot of closely spaced POIs, the aspect ratio has not been a problem in the finite element model.

To demonstrate the effect of changing the target aspect ratio, re-analyze the structure with 4 elements between girders and a target aspect ratio of 1.



3DFEM5 - Mesh Generation and Dead Load Analysis Example

With the target aspect ratio of 1, none of the elements have a length in the longitudinal direction greater than their width. In this case the target aspect ratio does not have a significant impact on the finite element model. For other structures the target aspect ratio could significantly impact the results. This model has 1817 total nodes, which is 12% more than the model generated with a target aspect ratio of 4.

The final option for the deck shell mesh generation is the node merge tolerance. By default, the tolerance is 0.1% of the span length. In previous versions of BrDR, before version 7.5, the tolerance could not be modified. New in version 7.5, the node merge tolerance can be modified for each span and can be input as a length or as a percent.

For this example, there are a lot of closely spaced shell elements created around the flange transitions. The default tolerance does not work well in this case because the span is short. For a longer span, such as 250 feet, this 0.1% tolerance would merge nodes within 3 inches. But for this short span, only 35 feet, the default tolerance is equivalent to less than 1/2".

Increase the node merge tolerance to avoid generating a lot of small shell elements. Modify the node merge tolerance to 1% of the span length. Use 4 elements between the girders and a target aspect ratio of 4.

When defining the node merge tolerance, the tolerance can be input as a length or percent of span. If the tolerance is input as a length, the program will convert the length to an equivalent percent of span using the span length.

Girder System Superstructure Definition		- 0	×
Definition Analysis Specs Engine Structural slab thickness Image: Structural slab thickness for rating Image: Structural slab thickness for rating Image: Consider structural slab thickness for rating Image: Structural slab thickness for design Image: Consider wearing surface for rating Image: Consider wearing surface for rating Image: Consider wearing surface for rating Image: Consider striped lanes for rating Default analysis type: Image: Longitudinal loading Vehicle increment: Image: Transverse loading Vehicle increment in lane: Image: Longitudinal loading Vehicle increment: Image: Longitudinal loading Vehicle increment in lane: Imag	Number of shell elements In the deck between girders In the web between flanges Slower Faster 10 9 8 7 6 5 4 3 2 1 Target aspect ratio for shell elements Image: securate Image: securate Faster Less accurate 1 1.5 2 2.5 3 3.5 4 3D FE node generation tolerance Percentage Length Tolerance (%) Span Length Tolerance (%) (%) (%)		
LFR: Model non-composite regions as non-composite LFD: Model non-composite regions as non-composite LRFR: Model non-composite regions as non-composite	Image: Strain	ly Can	el

3DFEM5 - Mesh Generation and Dead Load Analysis Example

The deck mesh does not have the small shell elements with this larger tolerance.



The analysis results are similar to the results with the smaller tolerance. This finite element model uses 1229 total nodes, 20% fewer than the corresponding model with a 0.1% merge tolerance.

rint				C+====				Dead	and Care			
Load	Actions			Composite	(long t	erm) (St	ane 2 🗸	Load	Case 1 - Memb	er Dist'd L		
	Actions			composite	(iong t		uger ·	Louid	cuse i meme			
Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)		
1	0.00	0.0	Right	-7.32	2.60	3.01	0.00	2.98	0.0000	0.0000		
1	3.41	10.0	Left	1.68	2.27	-2.91	0.00		0.0000	-0.0046		
1	3.41	10.0	Right	1.61	2.05	2.83	0.00		0.0000	-0.0046		
1	5.00	14.6	Left	4.69	2.06	-2.83	0.00		0.0000	-0.0065		
1	5.00	14.6	Right	4.58	1.76	2.73	0.00		0.0000	-0.0065		
1	6.83	20.0	Left	7.61	1.76	-2.73	0.00		0.0000	-0.0085		
1	6.83	20.0	Right	7.51	1.37	2.63	0.00		0.0000	-0.0085		
1	10.24	30.0	Left	11.62	1.37	-2.63	0.00		0.0000	-0.0116		
1	10.24	30.0	Right	11.69	1.10	2.50	0.00		0.0000	-0.0116		
1	13.66	40.0	Left	14.15	0.77	-2.34	0.00		0.0000	-0.0135		
1	13.66	40.0	Right	14.24	0.60	2.23	0.00		0.0000	-0.0135		
1	17.07	50.0	Left	15.12	0.37	-2.09	0.00		0.0000	-0.0142		
1	17.07	50.0	Right	15.38	0.06	1.97	0.00		0.0000	-0.0142		
1	18.99	55.6	Left	15.08	0.06	-1.97	0.00		0.0000	-0.0140		
1	18.99	55.6	Right	15.34	-0.20	1.88	0.00		0.0000	-0.0140		
1	20.48	60.0	Left	14.77	-0.20	-1.88	0.00		0.0000	-0.0135		
1	20.48	60.0	Right	14.98	-0.40	1.84	0.00		0.0000	-0.0135		
1	23.90	70.0	Left	12.83	-0.74	-1.80	0.00		0.0000	-0.0115		
1	23.90	70.0	Right	13.05	-0.96	1.79	0.00		0.0000	-0.0115		
1	27.31	80.0	Left	8.87	-1.29	-1.80	0.00		0.0000	-0.0084		
1	27.31	80.0	Right	9.40	-1.82	1.87	0.00		0.0000	-0.0084		
1	29.14	85.4	Left	6.14	-1.84	-1.87	0.00		0.0000	-0.0064		

3DFEM5 – Mesh Generation and Dead Load Analysis Example

Increasing the node merge tolerance can negatively impact the accuracy of the finite element model, especially if the tolerance is large. The merge tolerance essentially permits the finite element model to slightly differ from the input structure in order to generate a more regular grid of elements. In the above case, if the span length was 100 ft, the 1% merge tolerance would correspond to 1 ft. A finite element model with that tolerance may model diaphragm locations or flange transitions as far as 1 ft away from the actual distance defined along a member. When modifying the model generation tolerance, the FE model should be inspected to verify that the appropriate beam element properties are used along the lengths of the girders. Verify the input tolerance does not cause section changes to be skipped or produce a finite element model with geometry substantially different from the actual structure.

Once an acceptable finite element model is found, save the bridge to the database with the updated analysis options.

3DFEM5 - Mesh Generation and Dead Load Analysis Example



The live load analysis and spec checking can now be performed on this structure. The live load analysis and spec checking require more time to complete than the dead load analysis. Use the dead load only analysis first to find the best mesh generation options.