Release Note

Release Date : Feb., 2021

Product Ver. : nGen 2021 v1.1

Go to FREE TYIAL

INSTALLER DOWNLOAD

Contents

1. **Critical Region as per EC8 and NTC 2018** *03*
2. **Revit – nGen Link** *06*
3. **Mesh View Mode** *07*
4. **Customize Shortcut** *09*

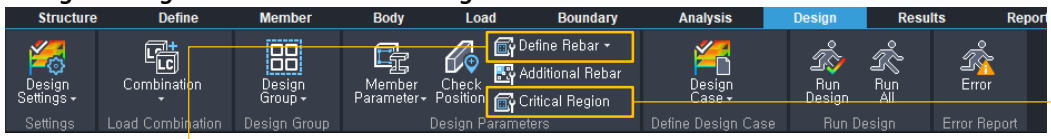
Go to FREE TYIAL

INSTALLER DOWNLOAD

1. Critical Region as per EC8 and NTC 2018

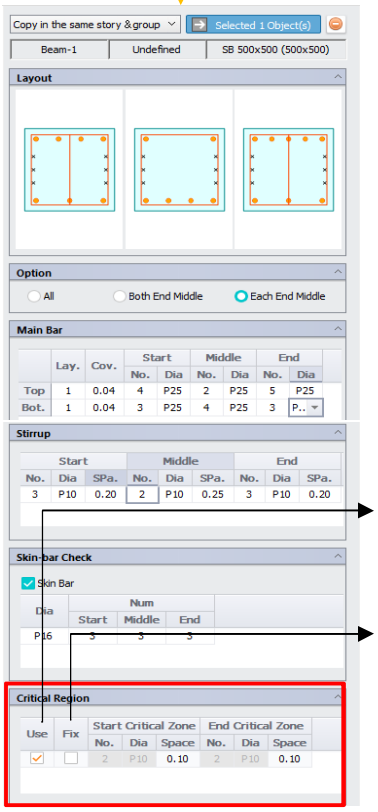
Setting & design for critical region in nGen

Design > Design Parameters > Critical Region



- Support a design for critical region in end zone of beam and column
- Support an auto-calculation of rebar spacing and length of critical region

Add Stirrup Information for critical region

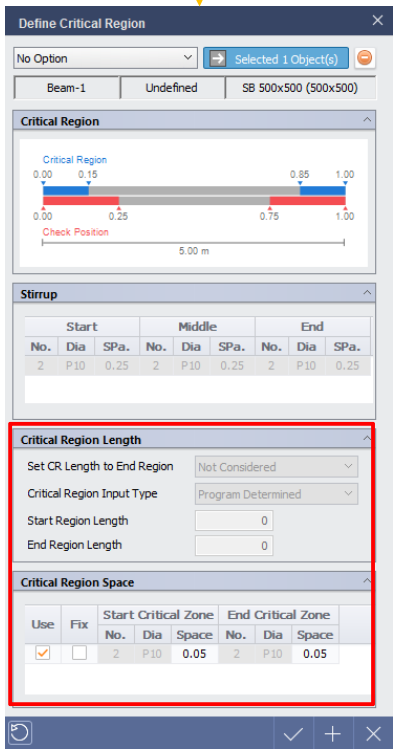


Set stirrup information in critical region

1. Use : It is activated when considering Seismic Design. When 'Use' is checked off, it is replaced with stirrup information.

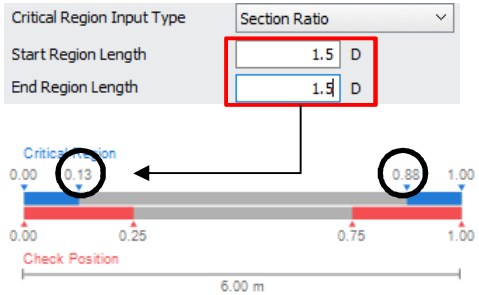
2. Fix : The entered information is fixed and never change during automatic design.

Add Critical Region Setting for each member



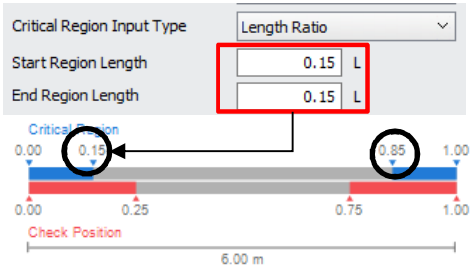
Set stirrup information & Application length in critical region.

[How to define critical region]



If D=0.5m,

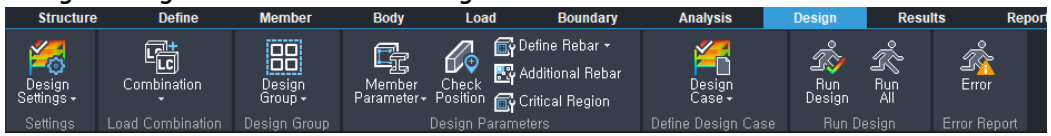
Critical Region = 1.5*0.5/L(=6.0m) = 0.75m → 0.125L



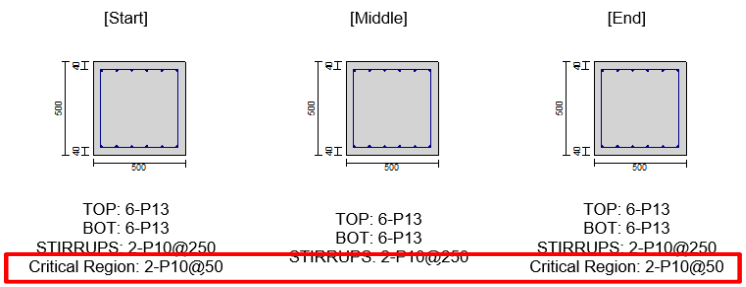
1. Critical Region as per EC8 and NTC 2018

Setting & design for critical region in nGen

Design > Design Parameters > Critical Region



Design Result for critical region



Add Rebar Data for Critical Region

1. ***Check Length of Critical Region***
2. ***Check Rebar Spacing of Critical Region***

F. Check Critical Region

Start. Length	Member	Beam-1	
	$L_{cr,min}, L_{cr,use}$	750.000 mm = 750.000 mm	OK
End. Length	Member	Beam-1	
	$L_{cr,min}, L_{cr,use}$	750.000 mm = 750.000 mm	OK
Start. Space	Member	Beam-1	
	$S_{cr,use}, S_{cr,max}$	50.000 mm < 78.000 mm	OK
End. Space	Member	Beam-1	
	$S_{cr,use}, S_{cr,max}$	50.000 mm < 78.000 mm	OK

Design Table for critical region

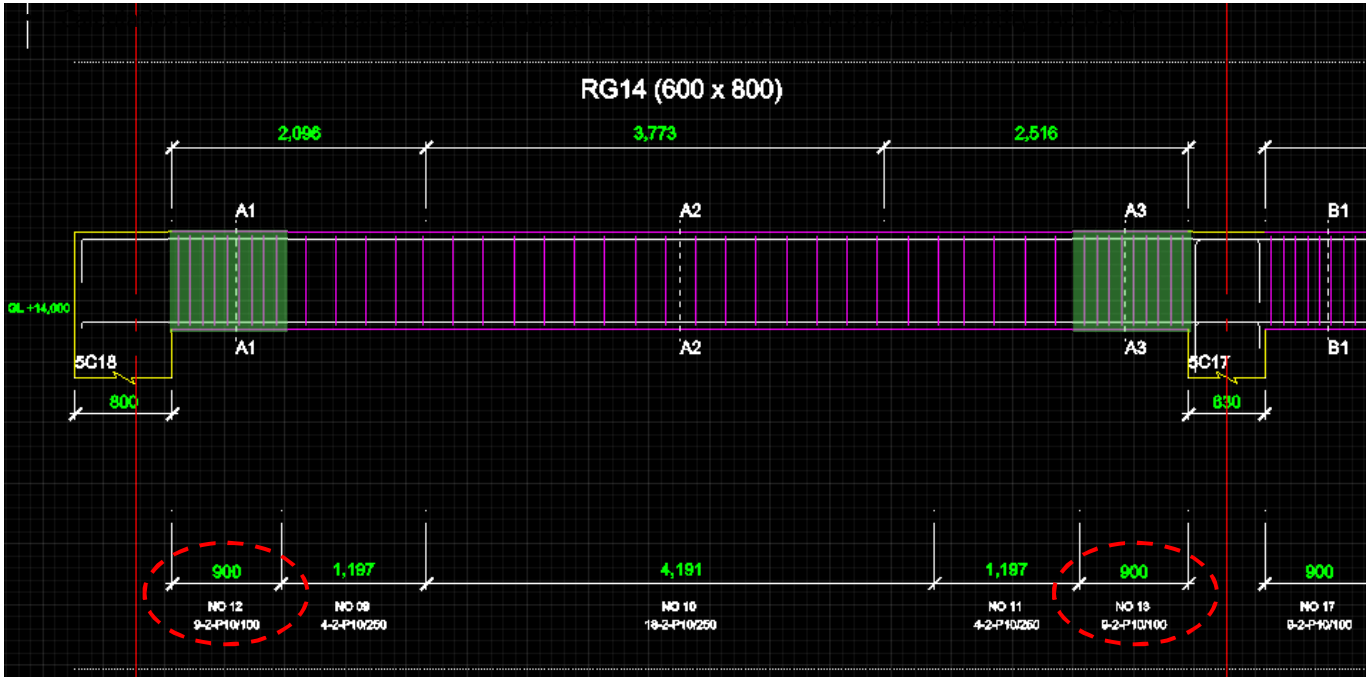
Check Rebar Spacing of Critical Region

	Status	Ratio		Story	Design Group		Member	Section			Material	Main Bar							Critical Region Smax (Start) ...			Critical Region Smax (End) (m)		
													Start		Middle		End		LCB		Use	LCB		Use
					Set	Group		Type	H (m)	B (m)		Posi.	Num	Dia	Num	Dia	Num	Dia	Position	-	Max	Position	-	Max
	NG	Main Re...	1.01	Roof	-	-	Beam-1	SB	0.50	0.50	C30/37	TOP	6	P13	6	P13	6	P13	fdbLCB1	OK	0.078	fdbLCB1	OK	0.078
	-	-	-									BOT	6	P13	6	P13	6	P13	-		0.078	-		0.078

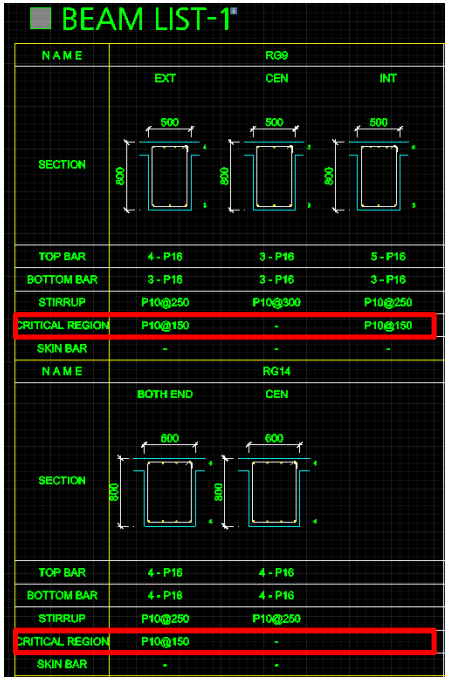
1. Critical Region as per EC8 and NTC 2018

Generation of critical region detail in midas Drawing

- Critical region detail is generated automatically in the beam and column drawing according to the critical region length and reinforcement spacing of nGen.
- Provide a rebar No. , dimension and area hatch in the critical region.
- Provides a rebar data for critical region to the beam-column list additionally.



Section Detail



BEAM LIST-1

NAME	RG9		
	EXT	CEN	INT
SECTION	500 / 800	500 / 800	500 / 800
TOP BAR	4 - P16	3 - P16	5 - P16
BOTTOM BAR	3 - P16	3 - P16	3 - P16
STIRRUP	P10@250	P10@300	P10@250
CRITICAL REGION	P10@150	-	P10@150
SKIN BAR	-	-	-

NAME	RG14	
	BOTH END	CEN
SECTION	600 / 800	600 / 800
TOP BAR	4 - P16	4 - P16
BOTTOM BAR	4 - P16	4 - P16
STIRRUP	P10@250	P10@250
CRITICAL REGION	P10@150	-
SKIN BAR	-	-

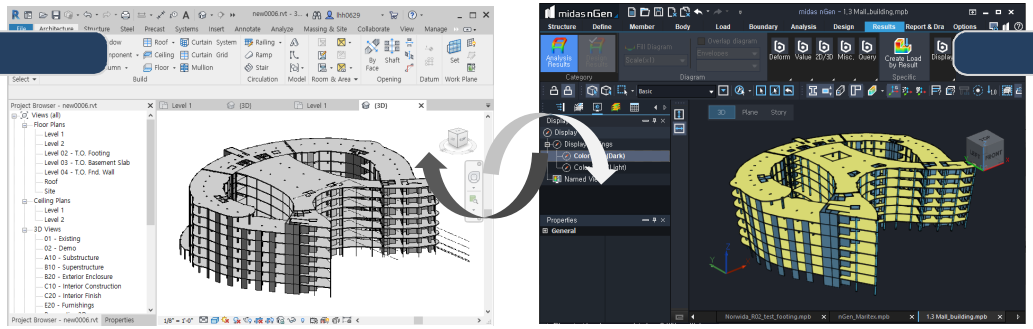
Section Detail

2. Revit – nGen Link

Supports a model conversion of nGen and Revit

Tutorial download

Revit Model



nGen Model

The following table shows the Revit components that are sent to midas nGen.

Components			Interface			Check
Properties	Grid		Revit	↔	nGen	Inclined and curved grids are not supported.
	Material		Revit	↔	nGen	
	Section		Revit	↔	nGen	
Member	Beam		Revit	↔	nGen	
	Column		Revit	↔	nGen	
	Wall		Revit	↔	nGen	
	Slab	Horizontal Slab	Revit	↔	nGen	
		Slope Slab	Revit	→	nGen	
		Drop Panel	Revit	→	nGen	
	Foundation	Mat footing	Revit	↔	nGen	
		Isolation footing	Revit	→	nGen	Import as mat footing member in midas nGen.
		Strip footing	Revit	→	nGen	Import as mat footing member in midas nGen.
	Brace		Revit	↔	nGen	
	Truss (Family)		Revit	→	nGen	Import as brace member in midas nGen.
	Opening		Revit	↔	nGen	
	Offset Information		Revit	↔	nGen	
Load				-		Not support
Boundary condition				-		Not support
Rebar Data				-		(nGen 2021 (v2.1))

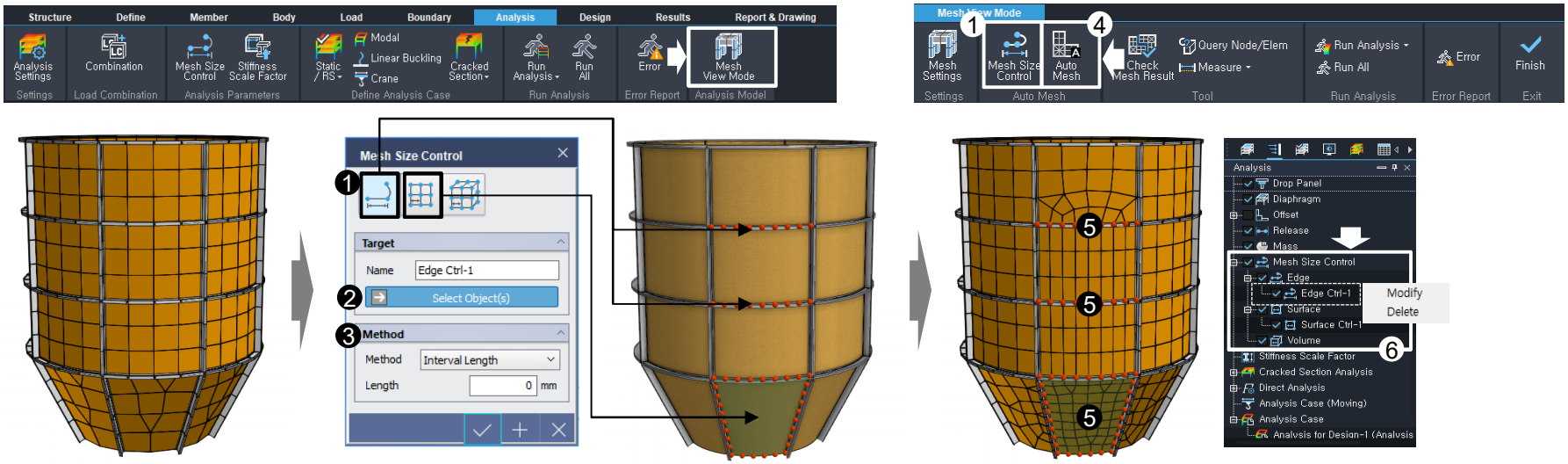
3. Mesh View Mode

Add Mesh View Mode

Analysis > Analysis Model > Mesh View Mode

- It takes a lot of time in checking mesh shape because the mesh shape can be confirmed after completing the analysis in the current version.
- When the desired shape is not generated by Auto-mesh or the wrong shape is generated, the mesh can be generated again by changing the mesh setting.
- "Check Mesh Result" support to find a mesh error before running an analysis.

Mesh Size Control in Mesh View Mode



❶ Select Element Type. (1D element or edge of 2D element, 2D, and 3D element)

❷ Select the target elements.

❸ Select the mesh method and value to divide.

❹ Run "Auto Mesh" (When the mesh control information are modified by "Mesh Setting" or "Mesh Size Control", Meshed shape can be displayed with "Auto Mesh" feature.)

❺ Check the meshed shape.

❻ To modify, Click Tree > Analysis tap > Mesh Size Control > Mesh name

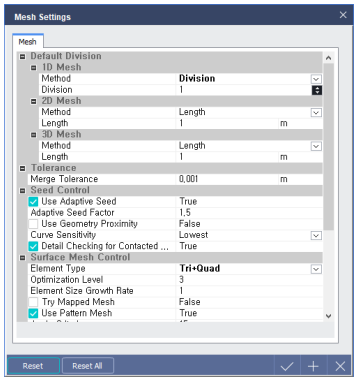
3. Mesh View Mode

Add Mesh View Mode

Analysis > Analysis Model > Mesh View Mode

[Mesh setting]

Global setting for Auto mesh



[Auto mesh]

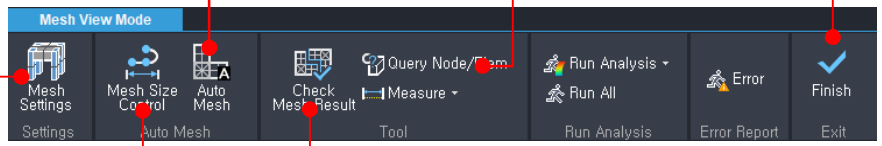
Run auto mesh
in mesh view mode

[Node/Element, Measure]

Show a node and elements No.

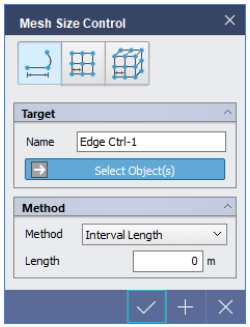
[Finish Mesh View Mode]

Finish and exit mesh view mode.



[Mesh size control]

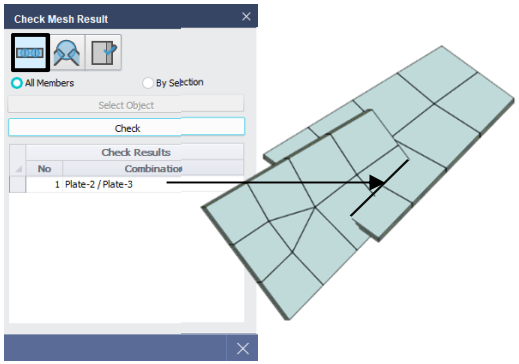
Mesh size and seeding
control for each members



[Check mesh]

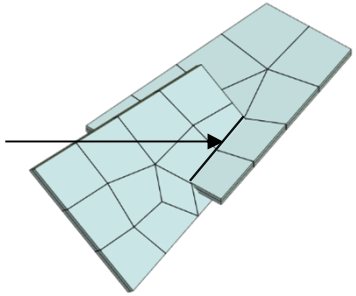
Check the result of auto mesh.
- Check unconnected members
- Check adjacent nodes
- Check sliver element

Incorrect mesh shape



the mesh shape of the two elements do not interfere with each other

Normal mesh shape

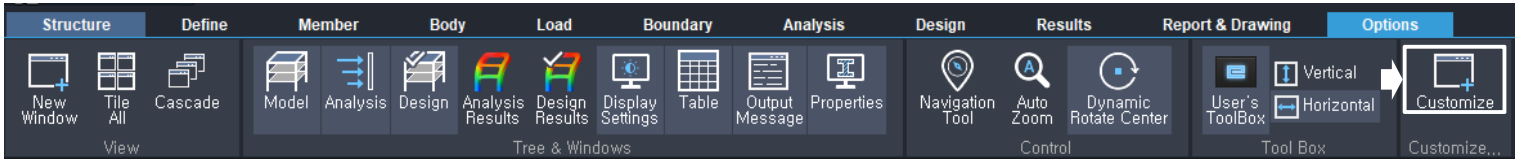


the mesh shape of two elements are generated to interfere with each other.

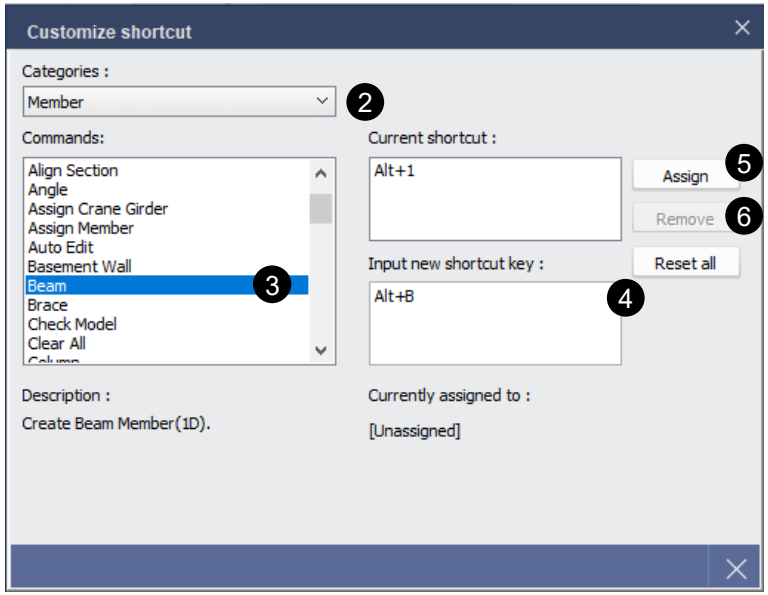
4. Customize Shortcut

Add Shortcut feature

Option > Customize > Customize



How to customize a shortcut key



❶ Click “Customize Shortcut” (Options > Customize Shortcut > Customize Shortcut)

❷ Select Categories of menu.

❸ Select Command to define shortcut key.

❹ Click in box of “Input new shortcut key” and execute the desired shortcut key.

❺ Click “Assign” button.

❻ To remove the existing shortcut key in “Current shortcut” box, Click “Remove” button.