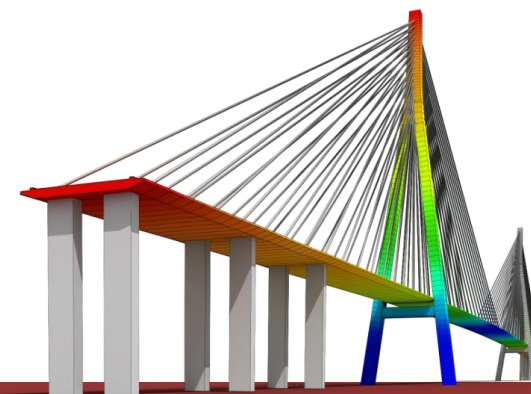




MIDAS *Technical
Material*

Tutorial

3-D Box Culvert using wizard



Specifications of the Bridge

Design of 3D Box Culvert 5.00 m X 3.50 m:

Clear Span of Box	= 5.00 m
Thickness of Side walls	= 0.35 m
Clear Height of the Box	= 3.50 m
Thickness of Deck and base Slabs	= 0.325 m
Idealized Span = $5000 + (350/2) + (350/2)$	= 5.35 m
Idealized Height = $(325/2) + 3500 + (350/2)$	= 3.8375 m
Clear Carriageway Width	= 8.75 m
Thickness of Crash Barriers	= 0.45 m
Width of structure $8750 + 2 \times 450$	= 9.65 m

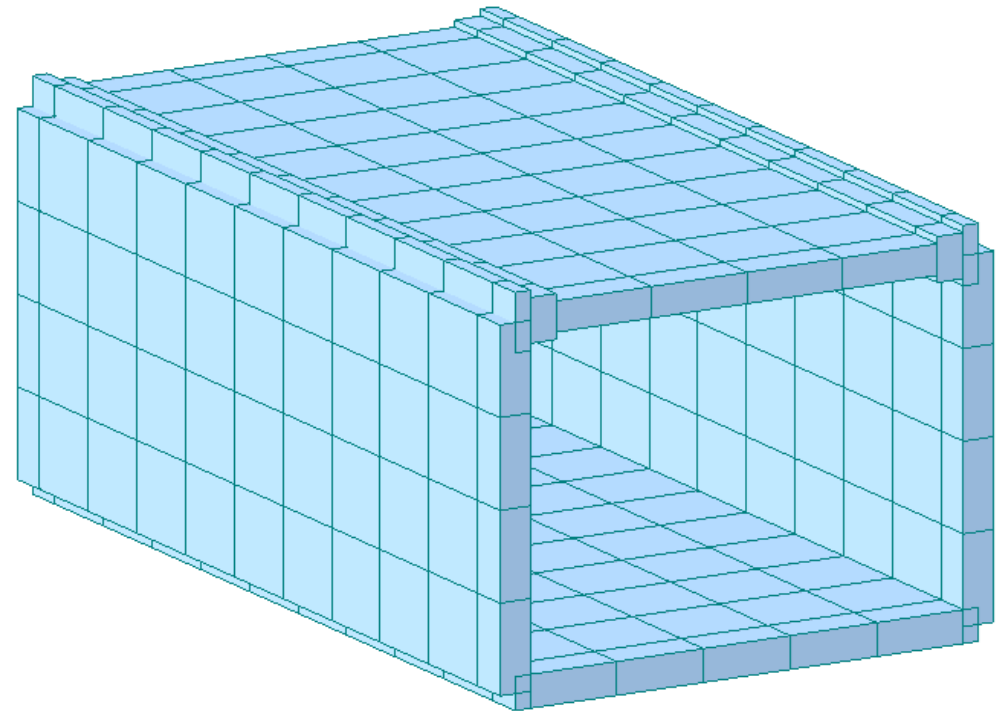


Table of Contents

1 - Model Generation

- Units Preference
- Definition of Properties
 - Material Properties
- RC Frame/Box Culvert Wizard:
 - Longitudinal
 - Transverse
 - Load
 - Generated Model
- Moving Load Generation
 - Activation of Slab Elements
 - Traffic Surface Lanes
 - Moving Load Case: Class A
 - Moving Load Case: Class 70R
 - EP LL Surcharge 2
- Domain Definition
 - Domain for Slab
 - Domain for Walls
 - Activation of Slabs
 - Sub-Domain for End Slabs
 - Sub-Domain for Mid Slabs
 - Activation of Walls
 - Sub-Domain for Top & Bottom Walls
 - Sub-Domain for Mid Walls

2 - Analysis & Result

- Perform Analysis
- Results
 - Auto Generation of LCB
 - Reactions
 - Displacement Contour
 - Plate Forces/Moments
 - Plate Cutting Diagram
 - Moving Load Tracer

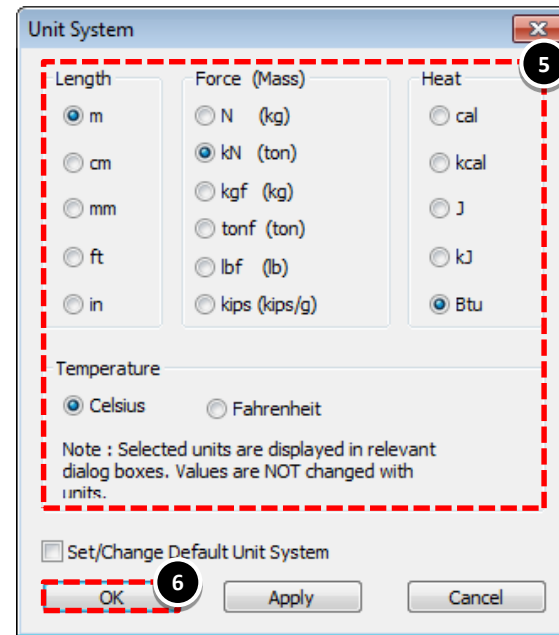
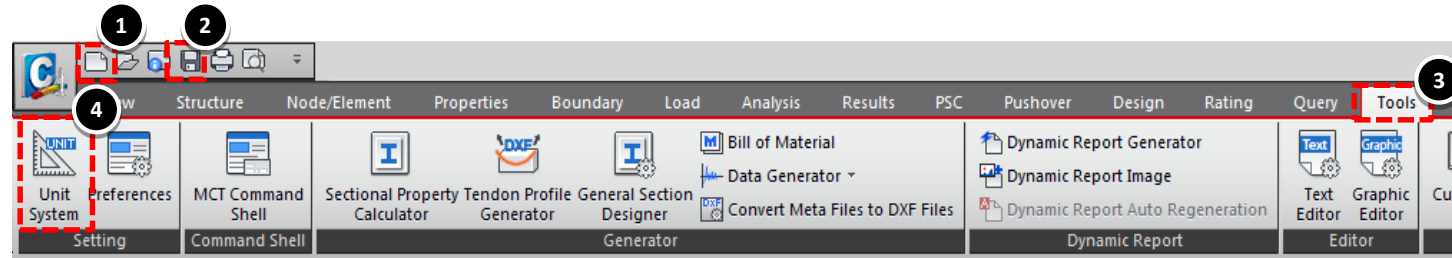
3 - Design

- Modify Concrete Material
- Plate Beam Data for Design
- Plate Column Data for Design
- Plate Beam Design
- Plate Column Design
- Rebar Input for Beam
 - End Slabs
 - Mid Slabs
- Rebar Input for Column
 - Top and Bottom Walls
 - Mid Walls
- Plate Beam Checking
- Plate Column Checking

Units Preference

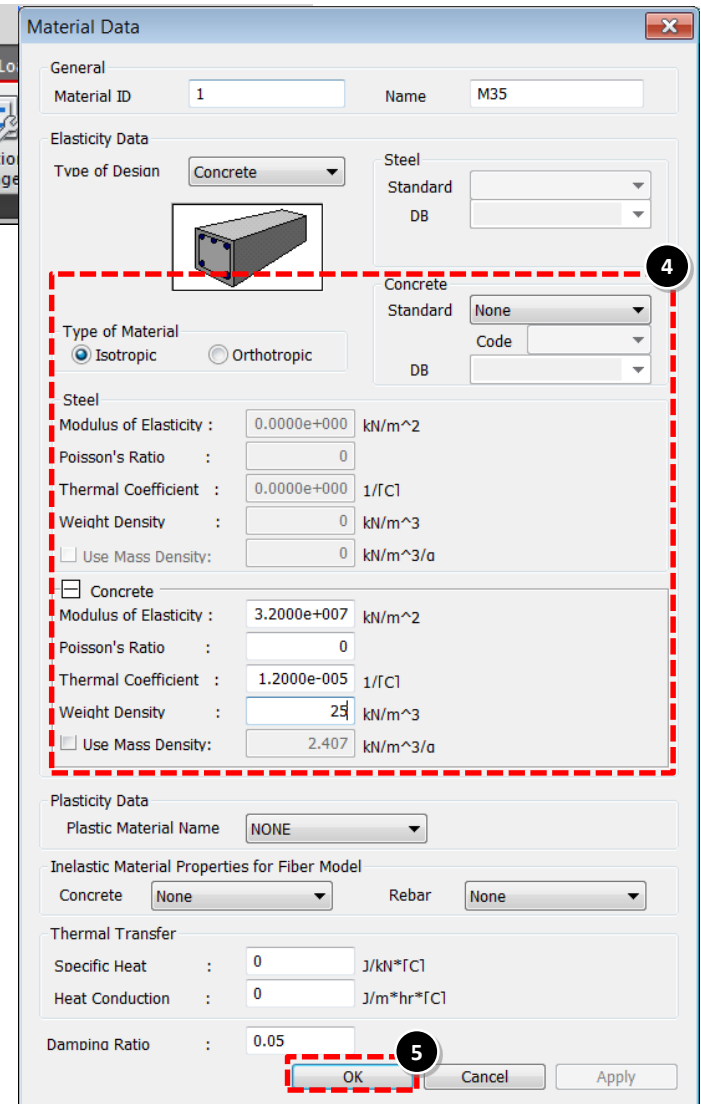
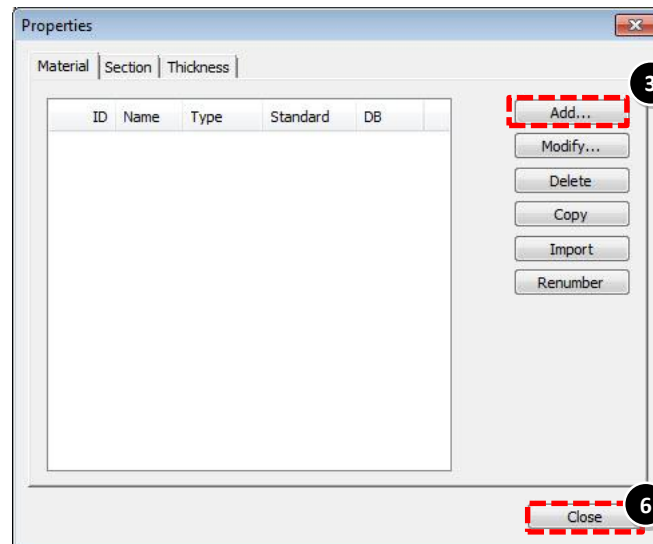
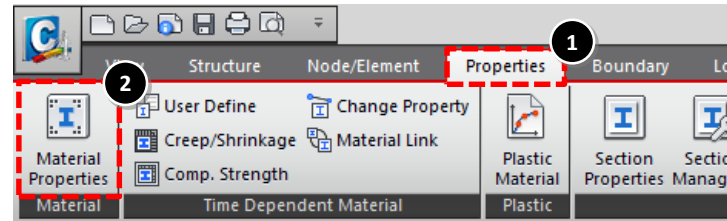
● Invoke midas Civil

- 1 Open New File
- 2 Save as “Box Culvert using wizard”
- 3 Go to > “Tools”
- 4 Click on “Unit System”
- 5 Select the unit system [m, kN(ton), Celcius]
- 6 Click on OK



Material Definition

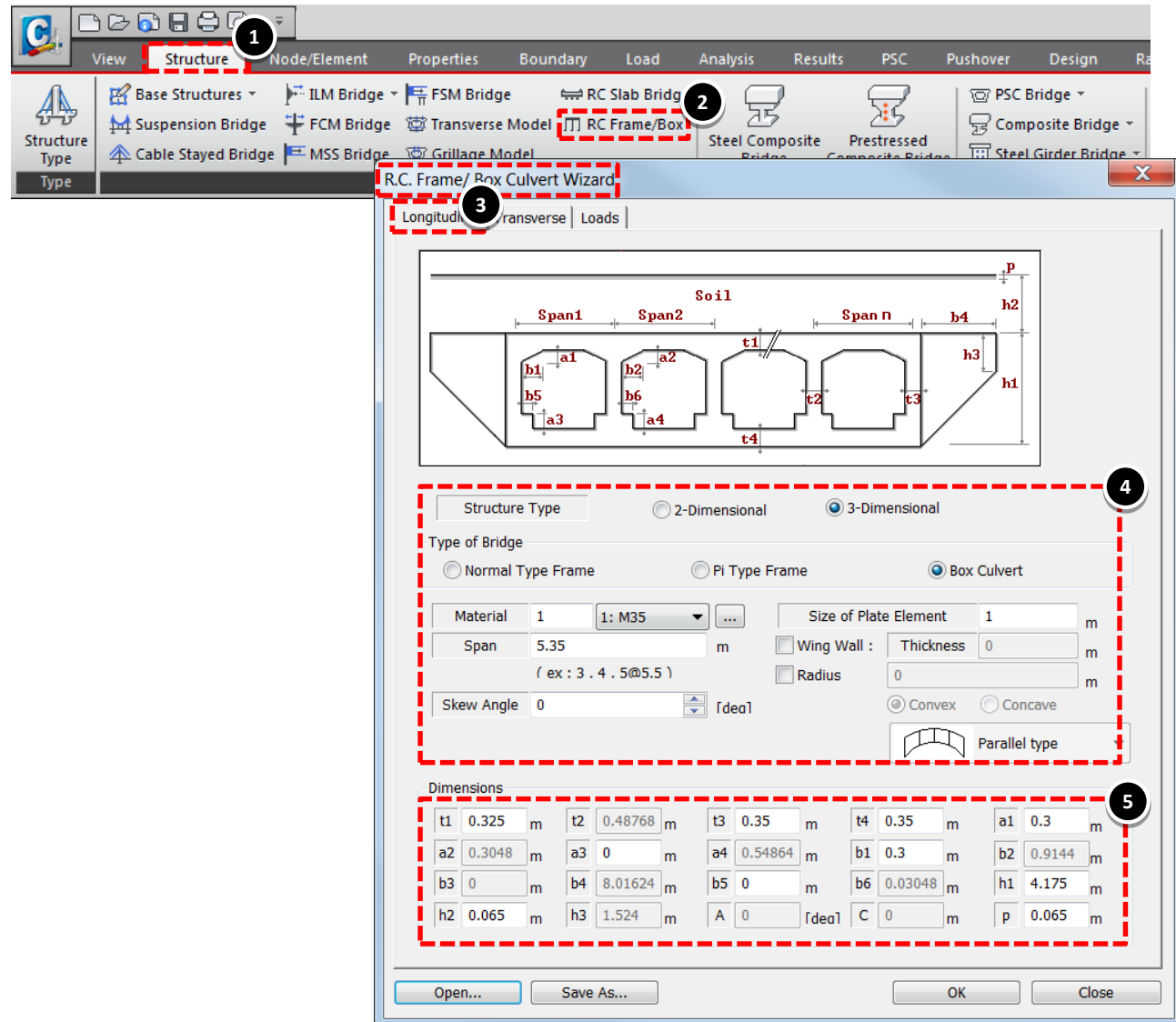
- 1 Go to > “Properties”
- 2 Click on “Material Properties”
- 3 Click on “Add” to define materials
- 4 Define Material data:
 - Name > M35
 - Type of design > Concrete
 - Concrete Standard > None
 - Modulus of Elasticity > $3.2\text{e}+007$
 - Poisson's ratio > 0
 - Thermal Coefficient > $1.2\text{e}-005$
 - Weight Density > 25
- 5 Click on OK
- 6 Click on Close



Note: When it is required to change some properties of material, the required database is selected and then standard is again set to None to alter the material Properties

Longitudinal Tab

- 1 Go to “Structure” tab
- 2 Click on “RC Frame/Box”
- 3 Go to “R.C.Frame/Box Culvert Wizard” tab
- 4 Define data:
 - structure type > “3-Dimensional”
 - Type of bridge > “Box Culvert”
 - Material > “M35”
 - Size of Plate Element > 1 m
 - Span > 5.35 m
 - Skew Angle > 0 (deg)
- 5 Define Dimension data:
 - Refer the drawing and enter the section dimension parameters
 - t1 : 0.325m, t3 : 0.35m, t4 : 0.35m, a1 : 0.3m, a3 : 0m, b1 : 0.3m, b5 : 0, h1 : 4.175m, h2 : 0.065m, p : 0.065m



Note: Click on 3 dots “...” icon and define the new material property.

Transverse Tab

- 1 Go to “Transverse” tab
- 2 Select Type > “Type 1”
Size of Plate Element > 1m
Refer the drawing and enter the section dimension parameters
b3 : 0.45m, b4 : 0m, b5 : 8.75m, b6 : 0m,
b7 : 0.45m
- 3 Define data for supports of culvert:
Spring Type > “General”
Modulus of Subgrade Reaction >
Lower > 7630 kN/m³
Length of Elastic Link > 1 m

R.C. Frame/ Box Culvert Wizard

Longitudinal | **Transverse** | Loads

Diagram showing cross-section dimensions: b3, b4, b5, b6, b7, D.

Type: Type1 | Size of Plate Element: 1 m

Parameters:

b1	0	m	b2	0	m	b3	0.45	m	b4	0	m
b5	8.75	m	b6	0	m	b7	0.45	m	D	6.096	m
n	0										

Supports of PI Frame

Transverse Fixed Support | from left side

Supports of Culvert

Spring Type: ☒ General ☐ Compression Only

Modulus of Subgrade Reaction

Lower	:	7630	kN/m ³
Lateral	:	19635.9	kN/m ³
Upper	:	19635.9	kN/m ³

Length of Elastic Link: 1 m

Buttons: Open... Save As... OK Close

Load Tab

- 1 Go to “Load” tab
- 2 Define Load Combination :
Select code > “IRC:6-2000”
Click > Define Moving Load Code
Select Moving Load Code > “India”
- 3 Click > “Self Weight”
Click > “Pavement”
Weight Density > 22 kN/m²
Thickness > 0.065m
Click > “Soil”
Weight Density > 20 kN/m²
Surcharge > 0
Phi > 30 (deg)
Click > “Barrier”
Self Weight > 11.25 kN/m
Additional Load > 0
Click > “Temperature Gradient”:
Delta T > 17.5 [T]
Click > Shrinkage Strain > 0.00019997
Thermal Coefficient > 1.2 e-005 1/[T]

- 4 If required for further use,
click on Save As... to save the
wizard file. This file saves
data input done in the wizard
- 5 Click on OK

R.C. Frame/ Box Culvert Wizard

Longitudinal | Transverse | **Loads**

Load Combinations : IRC:6-2000
Define Moving Load Code

☒ Self Weight

☒ Pavement : Weight Density 22 kN/m³ Thickness 0.065 m

☒ Soil : Weight Density 20 kN/m³ Phi 30 [deg]

Surcharge 0 kN/m²

Submerged Weight Density 10 kN/m³

Load Slope (L) 1 : 1.5 (R) 1 : 1.5

☐ Underground Water : GL - 0 m

☒ Barrier : Self Weight 11.25 kN/m Additional Load 0 kN/m

☐ Median Strip : 0 kN/m

☐ SideWalk : Weight Density 0 kN/m³ Thickness 0.3048 m

Crowd Load 0 kN/m²

☐ Live Load : Class of Vehicles ☒ Class A ☐ Class B ☒ Class 70R ☐ Class 40R

Eccentricity ☒ left ☐ right

☐ Settlement : 0 m

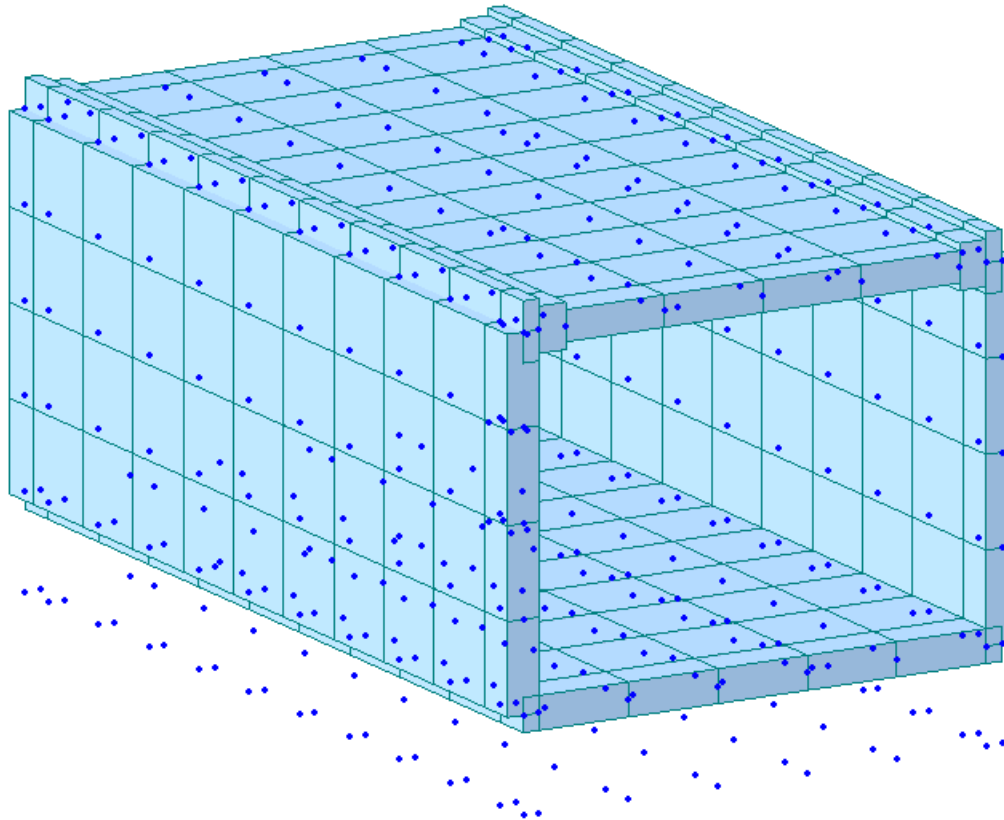
☐ System Temperature : T [+/-] 0 [T]

☒ Temperature Gradient : Delta T 17.5 [T]


☒ Shrinkage Strain : 0.00019997 Thermal Coefficient : 1.2e-005 1/[T]

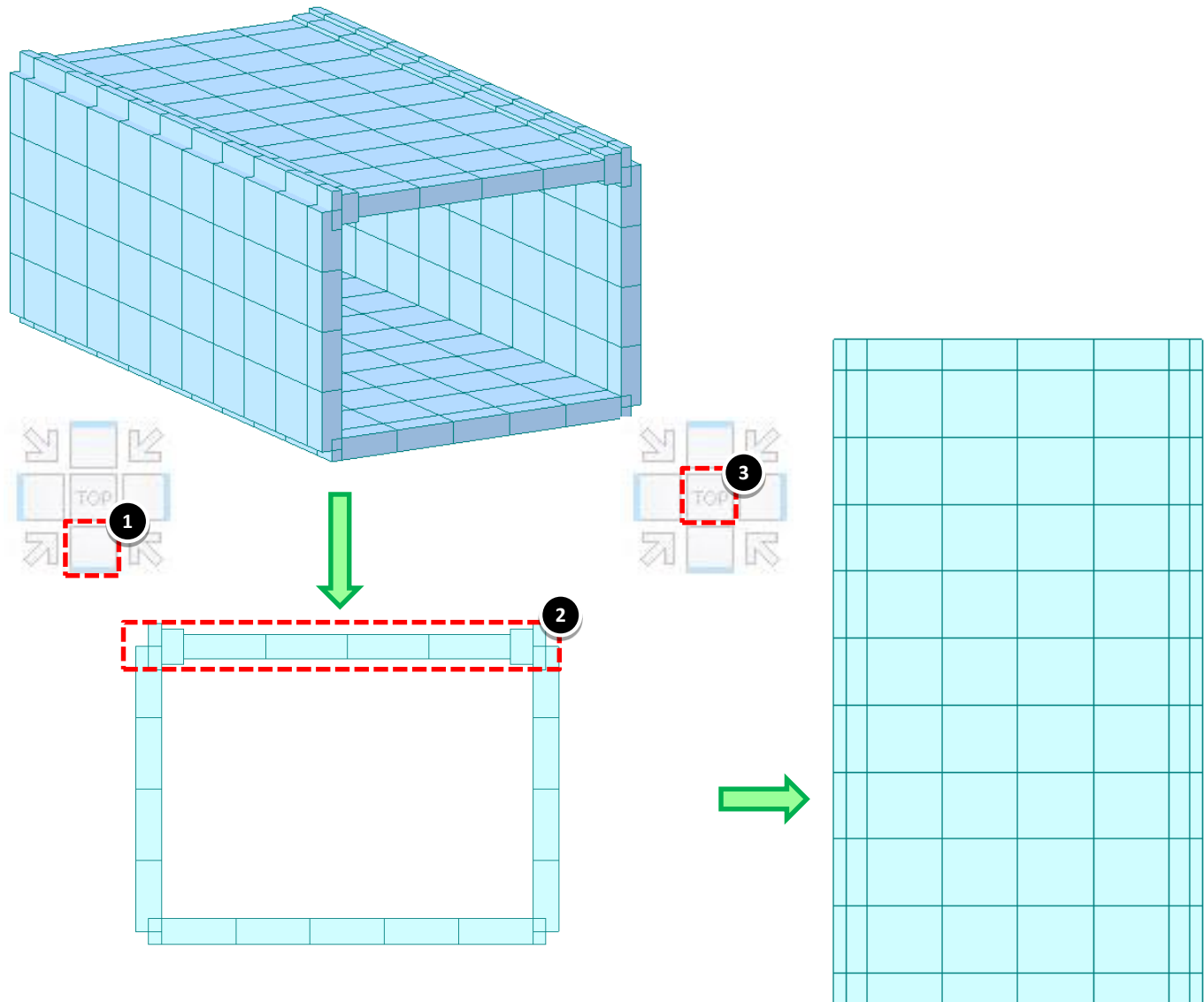
Open... Save As... OK Close

Note: Assuming area of crash barriers as 0.45 m² SW of CB = 0.45X 25 = 11.25 kN/m

Generated Model

Activation of Slab Elements

- 1 To view the model in Front view
Click on
- 2 Click on “**Select Single**” and
Select the top portion as shown
- 3 Click on “**Activate**” button
To view the model in top view
Click on
Click on “**Display**  > **Node** >
Uncheck the node option to
undisplay the nodes



Traffic Surface Lanes

- 1 Go to “Load” > “Moving Load”
- 2 Click “Traffic Surface Lanes”
- 3 Click “Add”
- 4 Lane Name > “Lane 1-Class A”
- 5 Lane Width “1.8”m
Wheel Spacing “1.8”m
Offset Distance to Lane Center “-1.75”m
Span Length > 5.35 m
- 6 Moving Direction > Both
Select Selection by “2 Points”
Click on node no. 1
Click on Node no. 97
- 7 Click “OK”
Similarly define remaining lanes as per the table

The screenshot displays the software interface for defining traffic surface lanes. The 'Load' menu is open, and the 'Moving Load' option is selected. The 'Define Design Traffic Surface Lane' dialog box is shown, with the following details:

- Lane Name:** Lane 1-Class A
- Traffic Lane Properties:**
 - Lane Width: 1.8 m
 - Wheel Spacing: 1.8 m
 - Offset Distance to Lane Center: -1.75 m
 - Span Length: 5.35 m
- Moving Direction:** Both
- Selection by:** 2 Points
- Selection points:** 0, 0, 0 m and 5.35, 0, 0 m

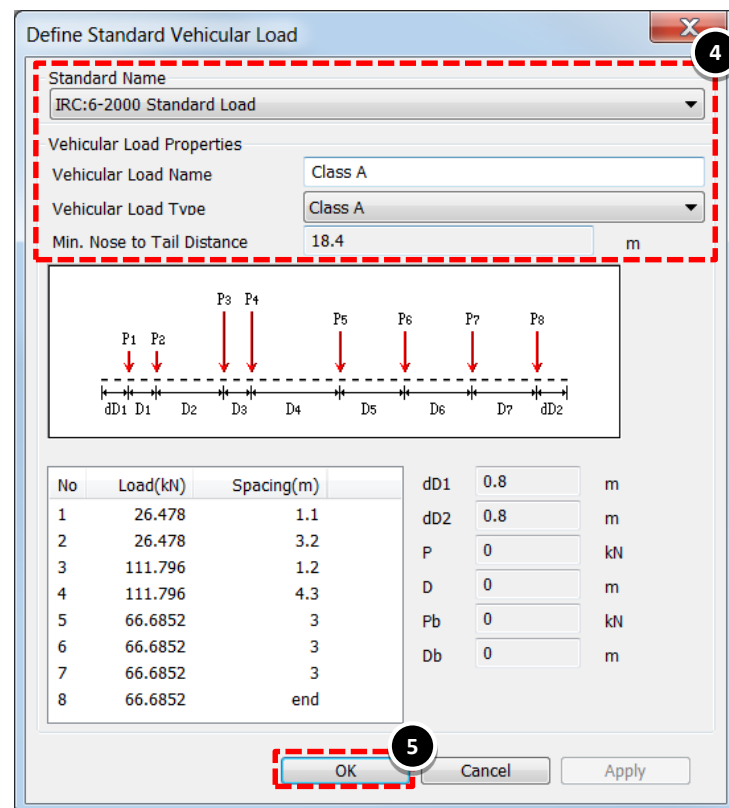
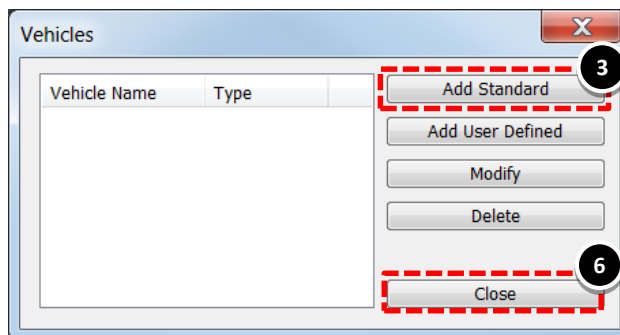
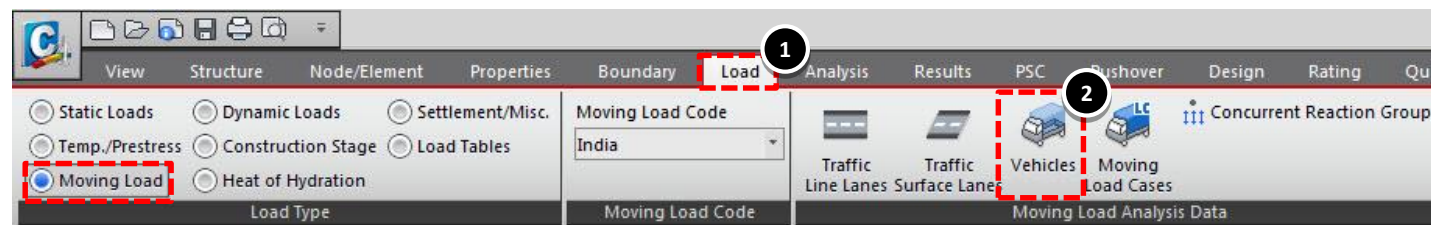
A table of lane data is provided below:

Lane Name	Lane Width	Wheel Spacing	Offset Distance
Lane 1-ClassA	1.8m	1.8m	-1.75m
Lane 2-ClassA	1.8m	1.8m	-5.25m
Lane 3-Class70R	1.93m	1.93m	-4.375m

The diagram shows a grid of nodes with nodes 1 and 97 highlighted, indicating the selection points for the lane definition.

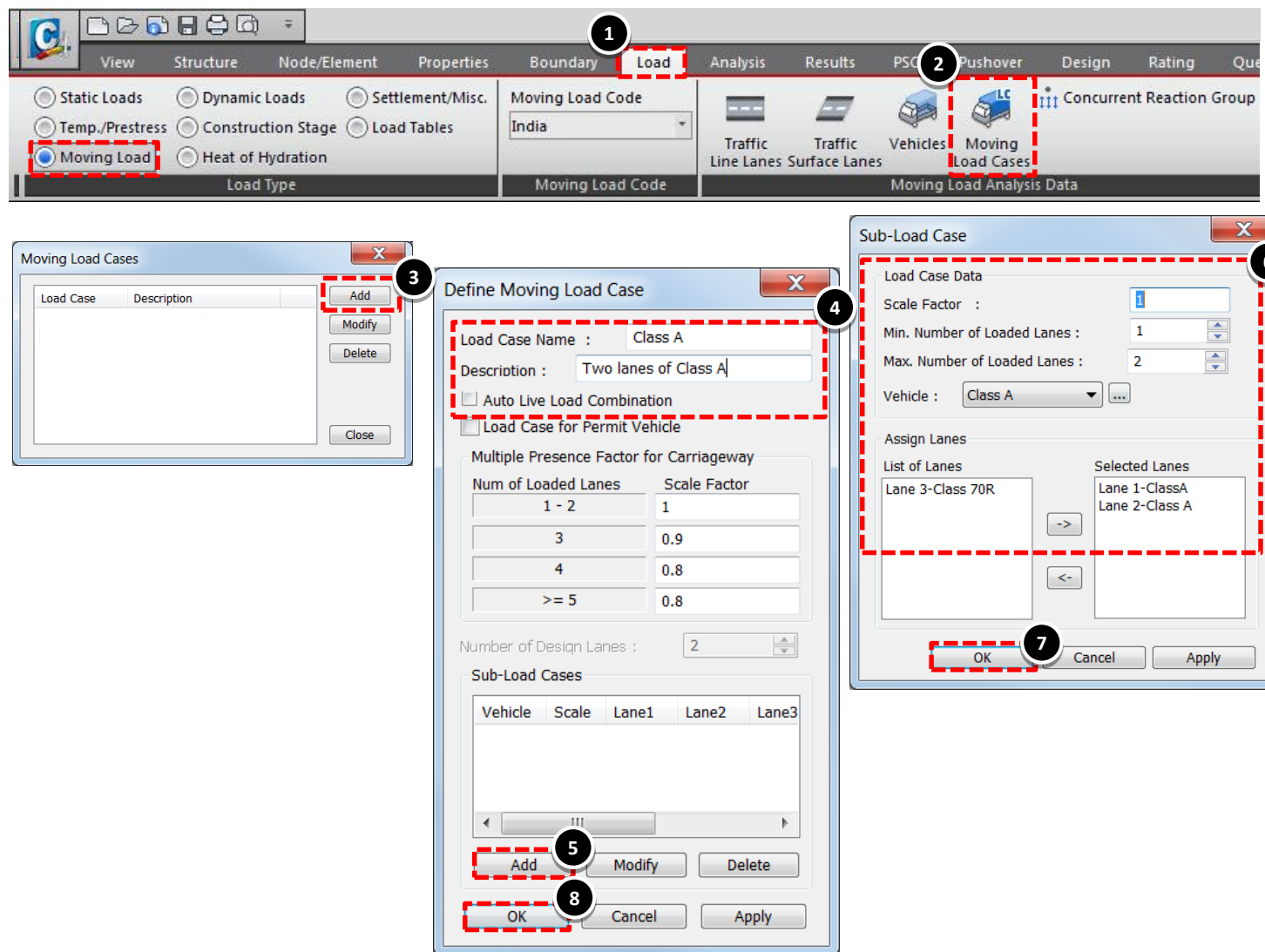
Define Vehicles

- 1 Go to “Load” > “Moving Load”
- 2 Click “Vehicles”
- 3 Click “Add Standard”
- 4 Select “Standard Name” > “IRC:6-2000 Standard Load”
 - Select Vehicular Load Type > “Class A”
 - Click “Apply”
- Similarly define vehicle standards as Class 70R
- 5 Click “OK”
- 6 Click “Close”



Moving Load Case: Class A

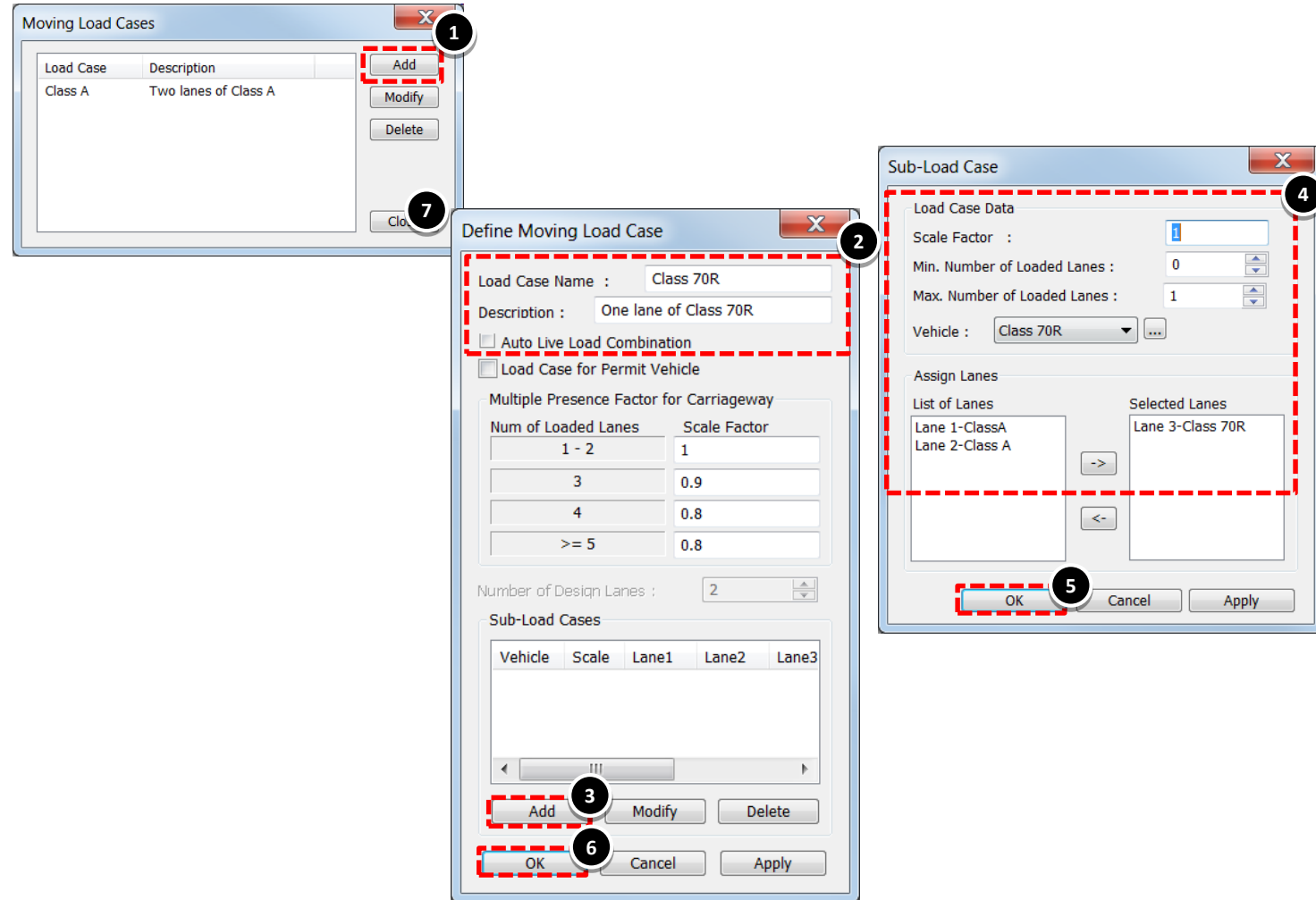
- 1 Go to “Load” > “Moving Load”
- 2 Click “Moving Load Cases”
- 3 Click “Add”
- 4 Enter Load Case Name “Class A”
Description > **Two lanes of Class A**
A
Uncheck “Auto Live Load Combination”
- 5 Click “Add” under Sub-Load Cases
- 6 In Sub-Load Case
Enter “Scale Factor” > 1
Enter “Minimum Loaded Lanes” > 1
Enter “Maximum Loaded Lanes” > 2
Select “Vehicle” > Class A
Under List of Lanes Select “Lane1- Class A” and “Lane2- Class A”
Click on “ ”
- 7 Click “OK”
- 8 Click “OK”



Note: To take into account of the wheel spacing and minimum clearance for different vehicles, the Auto Live Load Combination option can be unchecked and manually different moving load combinations can be created for the vehicles as per IRC 6:2000.

Moving Load Case-70R

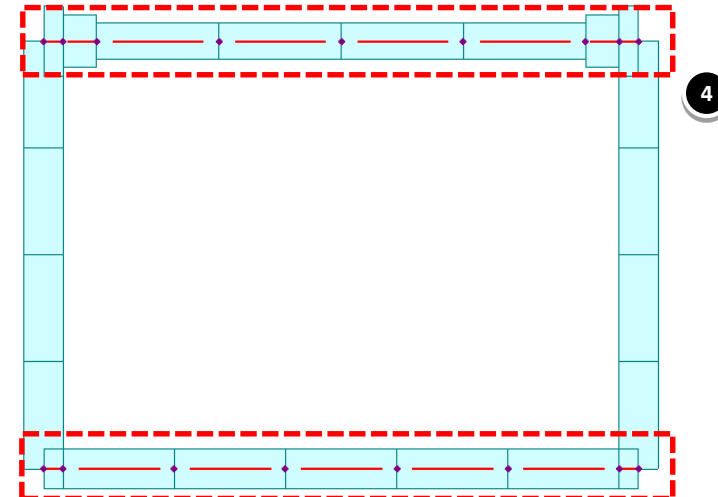
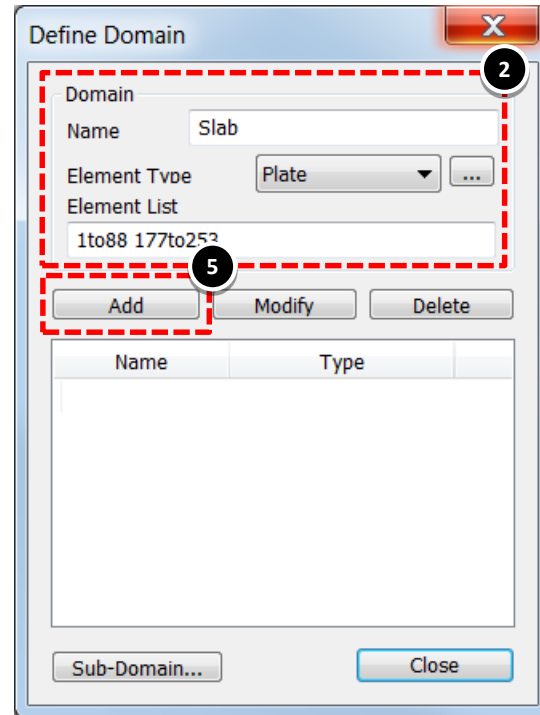
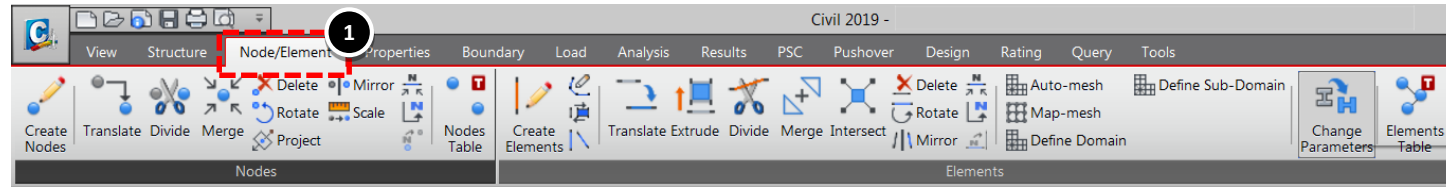
- 1 Click “Add”
- 2 Enter Load Case Name “Class 70R”
Description> **One lane of Class 70R**
Uncheck “Auto Live Load Combination”
- 3 Click “Add” under Sub-Load Cases
- 4 In Sub-Load Case
Enter “Scale Factor” > 1
Enter “Minimum Loaded Lanes” > 1
Enter “Maximum Loaded Lanes” >2
Select “Vehicle” > Class A
Under List of Lanes Select “Lane1-
Class A” and “Lane2- Class A”
Click on “>”
- 5 Click “OK”
- 6 Click “OK”
- 7 Click “Close”




Note: To take into account of the wheel spacing and minimum clearance for different vehicles, the Auto Live Load Combination option can be unchecked and manually different moving load combinations can be created for the vehicles as per IRC 6:2000.

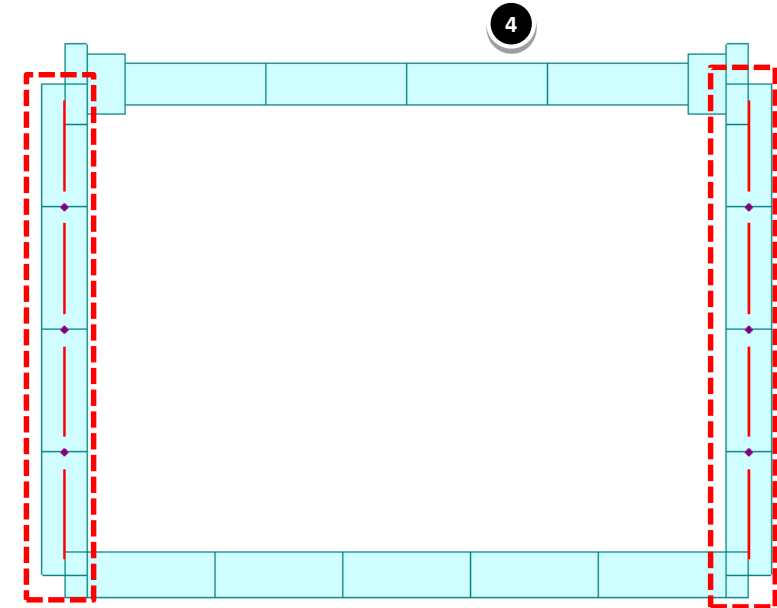
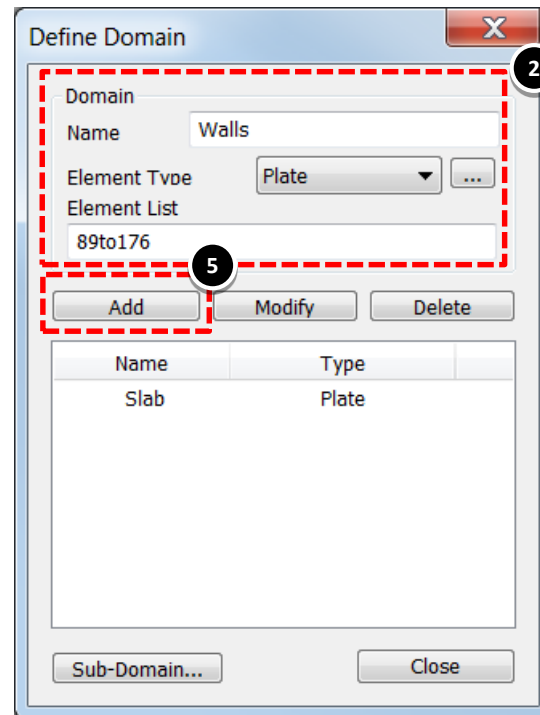
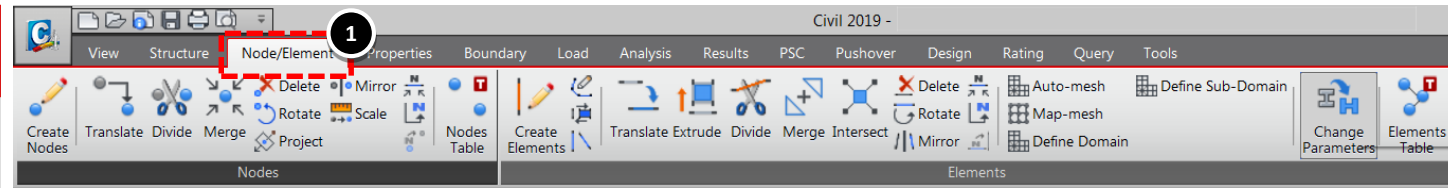
Domain for Slab

- 1 Go to Node/Elements > Define Domain
- 2 Domain Name > **Slabs**
- 3 Click on > Select Single
- 4 Select top and bottom slab Elements as in image
- 5 Click on > **Add**



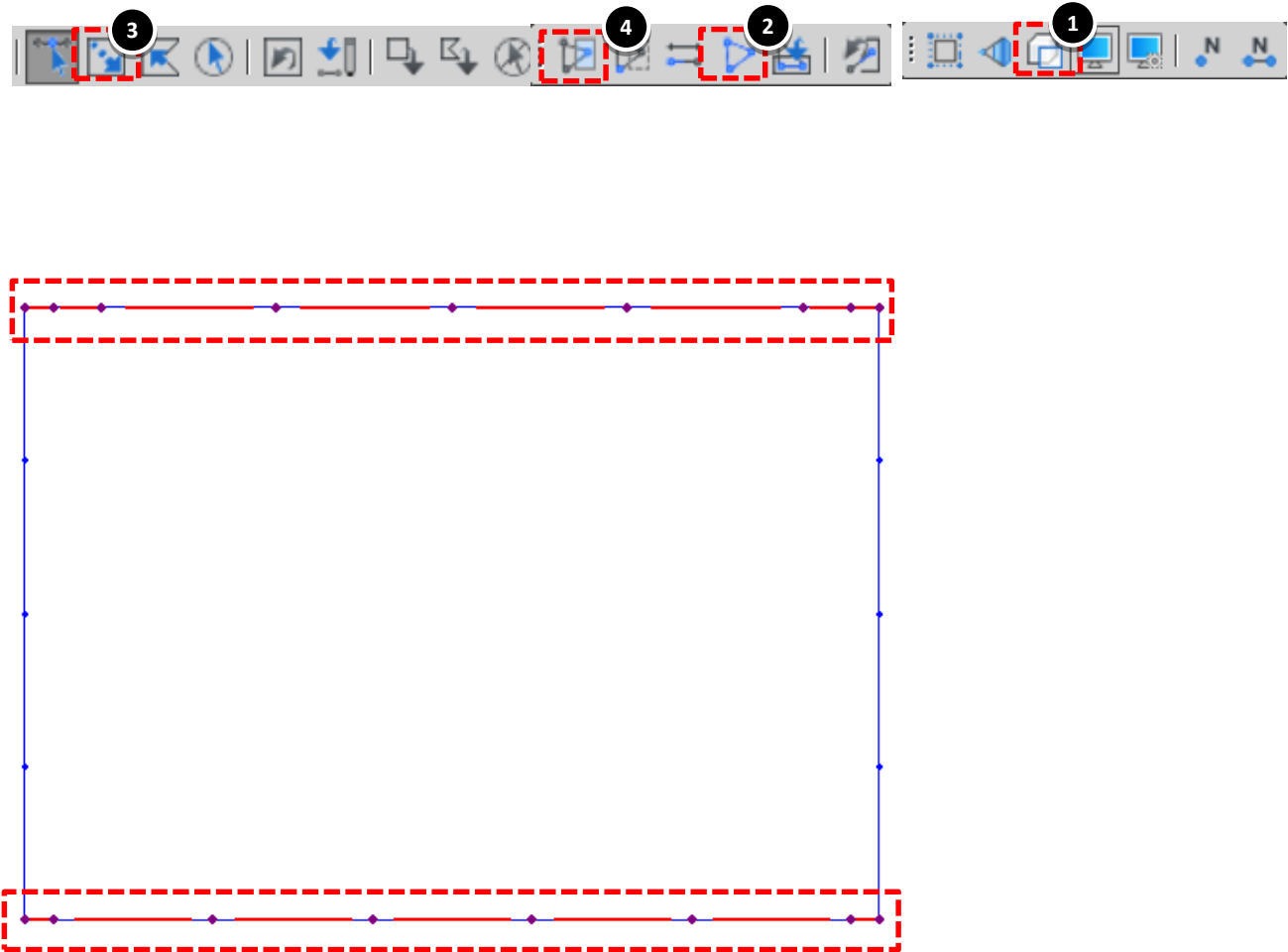
Domain for Walls

- 1 Go to Node/Elements > Define Domain
 - 2 Domain Name > **Side walls**
 - 3 Click on > Select Single
 - 4 Select top and bottom slab Elements as in image
 - 5 Click on > **Add**
- Click on “**Display**  > **Node** > **Uncheck the node option** to display the nodes



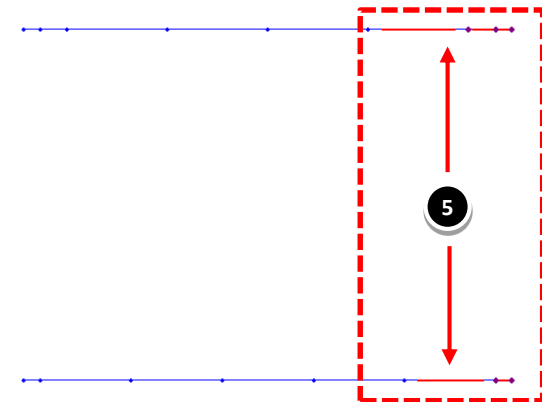
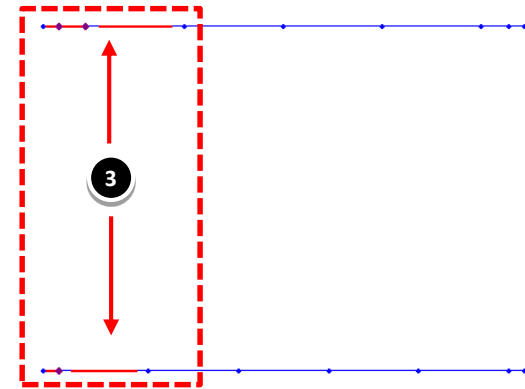
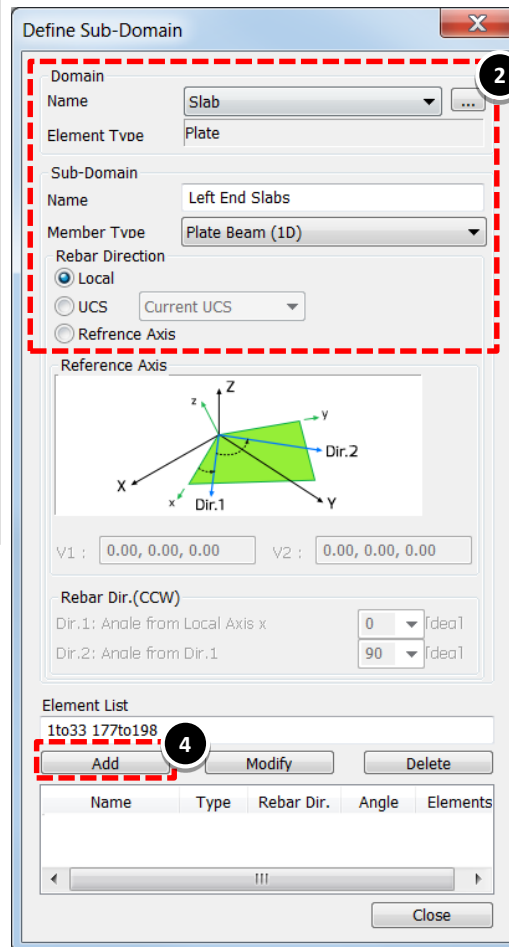
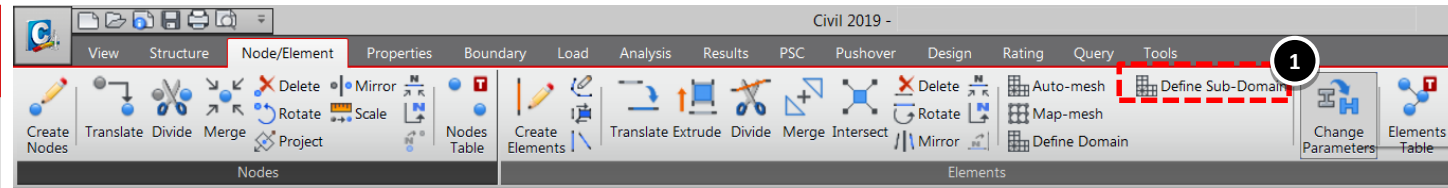
Activation of Slabs

- 1 Click on Hidden
- 2 Click on **Activate All**
- 3 Click on > **Select Single**
- 3 Select Side wall Elements as in image and click on activate



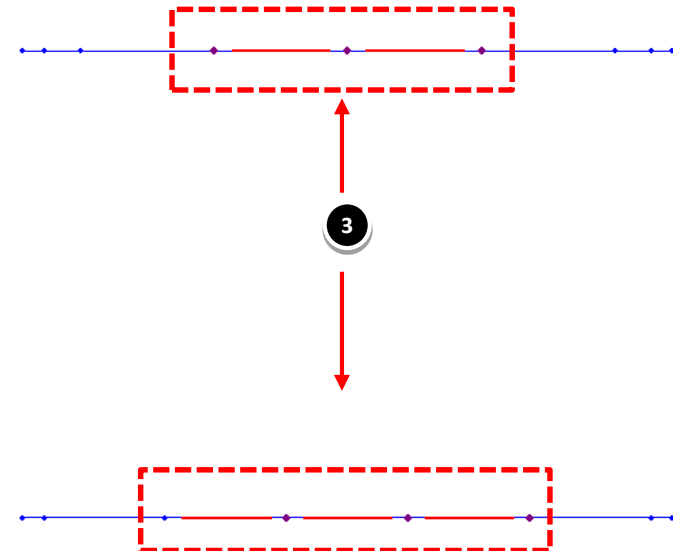
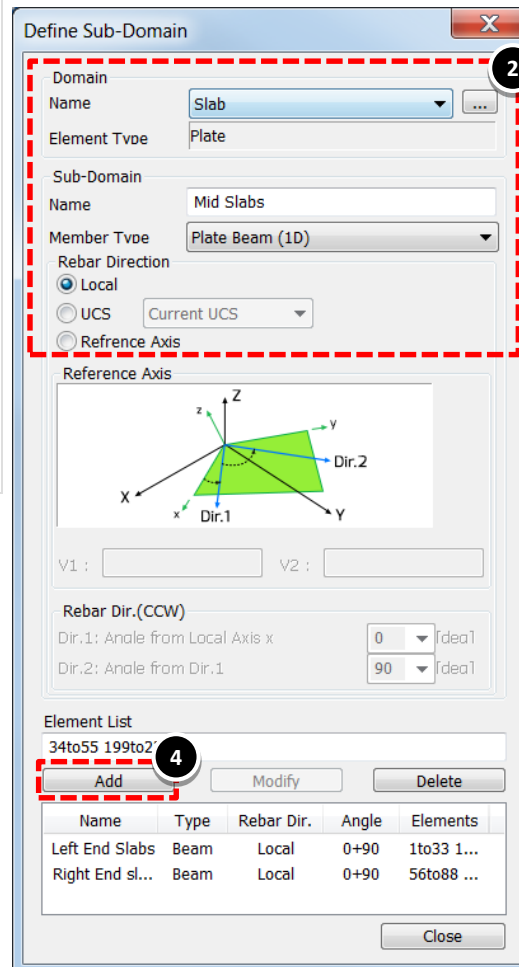
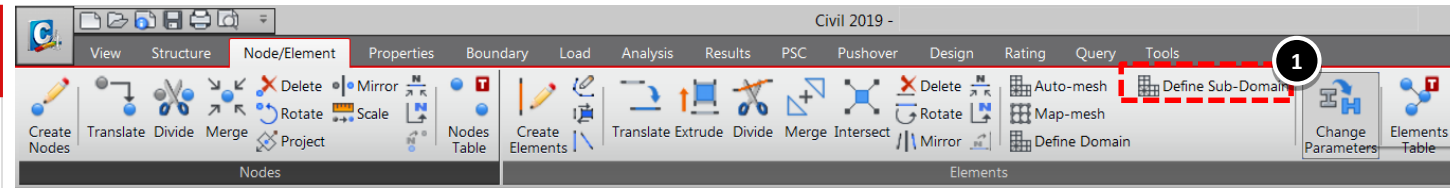
Sub-Domain for End slabs

- 1 Go to Node/Elements > Define Sub-Domain
- 2 Domain Name > **Slabs**
 Sub Domain Name > **Left End Slabs**
 Member Type > **Plate Beam (1D)**
 Rebar Direction > **Local**
- 3 Select Left end Elements as in image
- 4 Click on > **Add**
- 5 Repeat the **steps 1 to 5** to add **Right End Slabs**



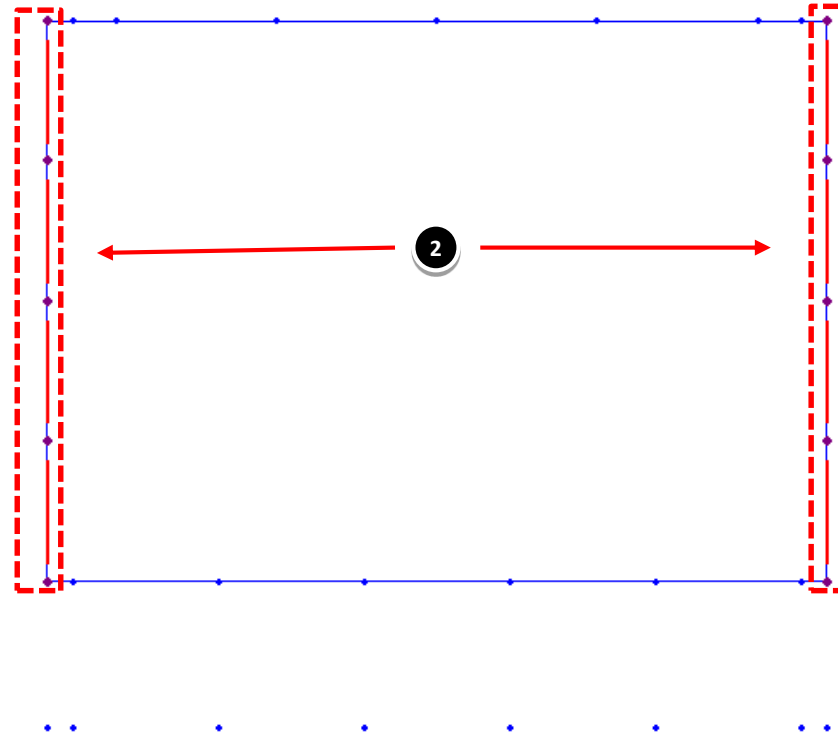
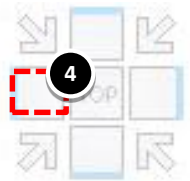
Sub-Domain for Mid slabs

- 1 Go to Node/Elements > Define Sub-Domain
- 2 Domain Name > **Slabs**
 Sub Domain Name > **Mid Slabs**
 Member Type > **Plate Beam (1D)**
 Rebar Direction > **Local**
- 3 Select top and bottom slab Elements as in image
- 4 Click on > **Add**



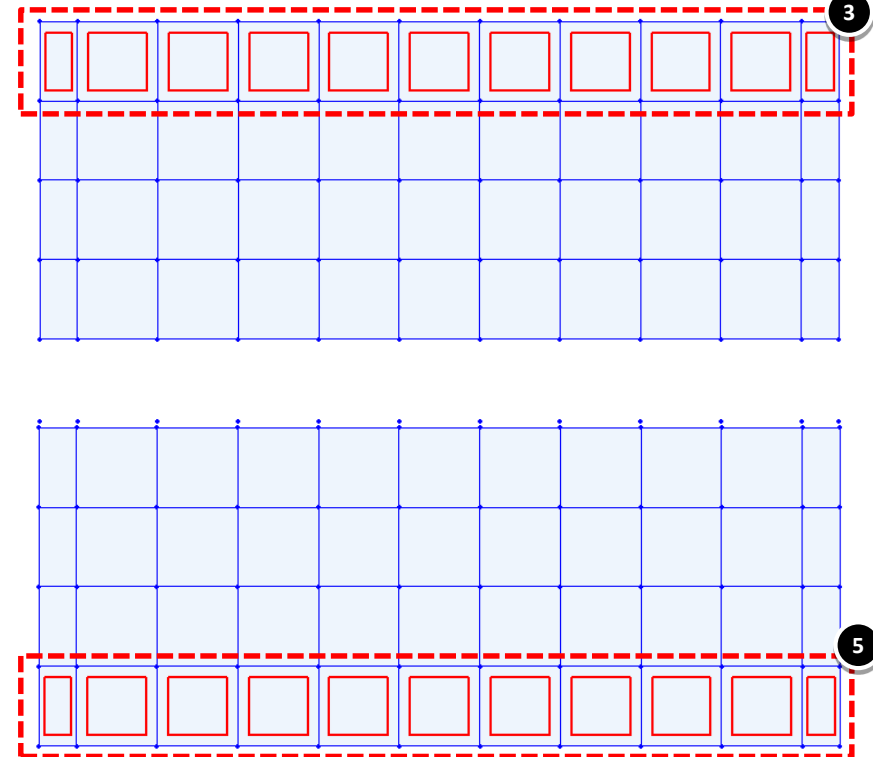
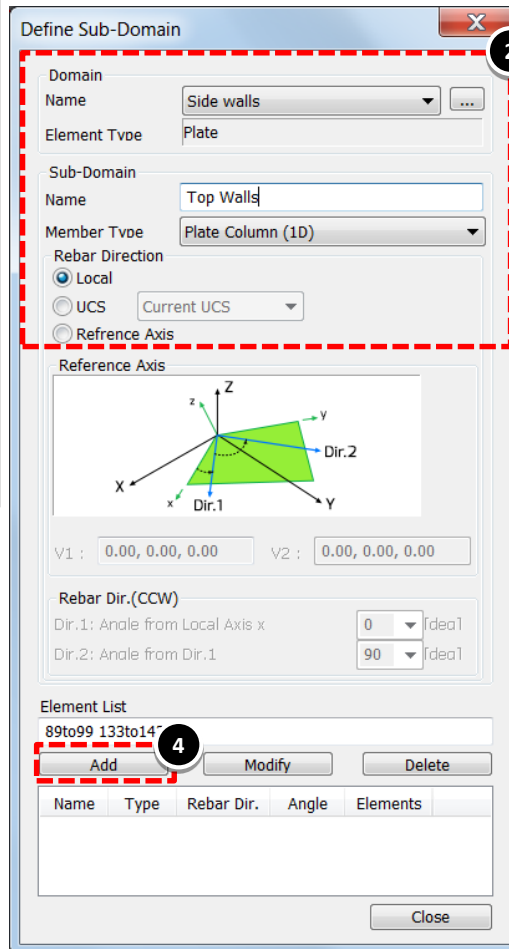
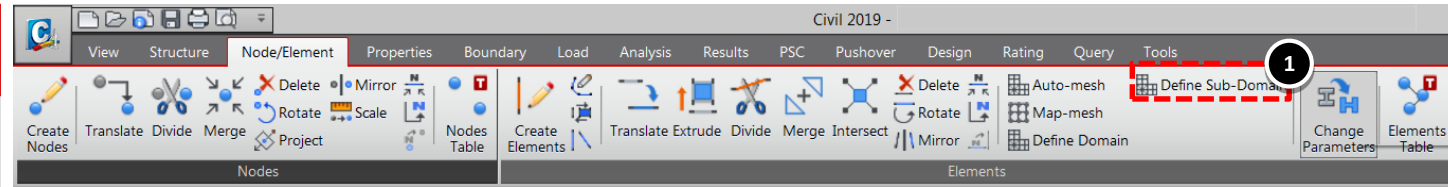
Activation of Walls

- 1 Click on **Activate All**
- 2 Select Side wall Elements(8to10 and 11to13) as in image
- 3 Click on **Activate**
- 4 Go to Side View



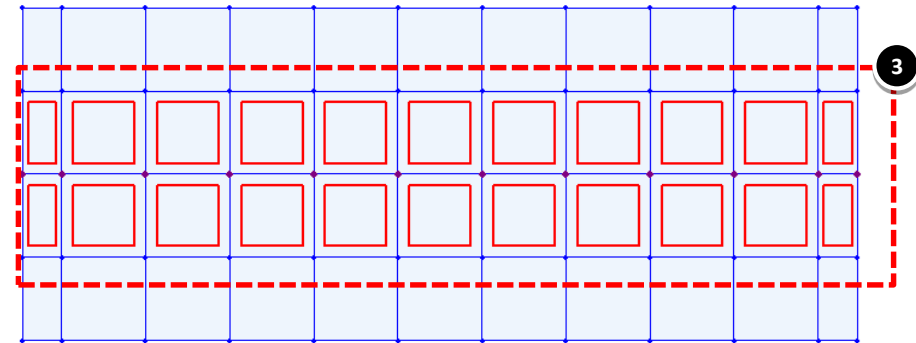
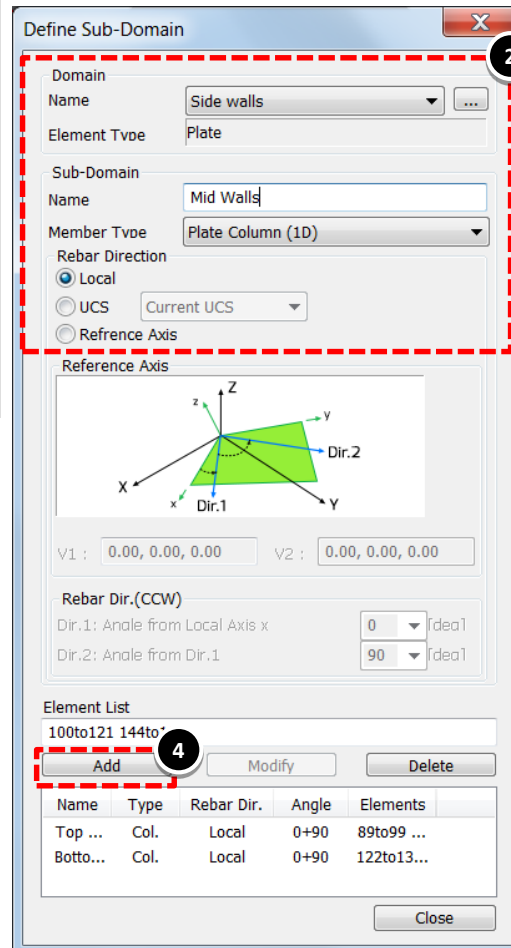
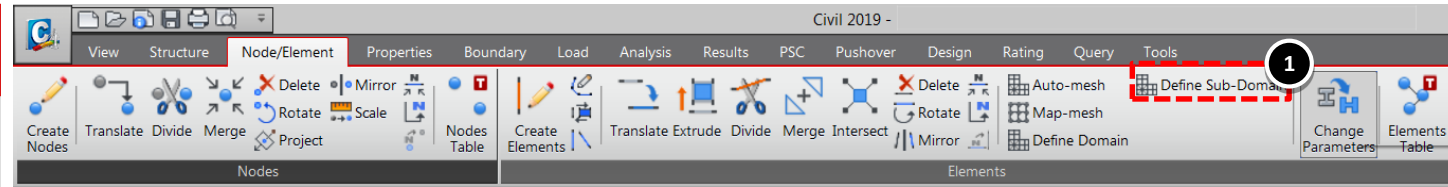
Sub-Domain for Top & Bottom walls

- 1 Go to Node/Elements > Define Sub-Domain
- 2 Domain Name > **Side walls**
 Sub Domain Name > **Top walls**
 Member Type > **Plate Column (1D)**
 Rebar Direction > **Local**
- 3 Select top wall Elements as in image
- 4 Click on > **Add**
- 5 Repeat **Steps 4 to 6** to add **Bottom Walls**



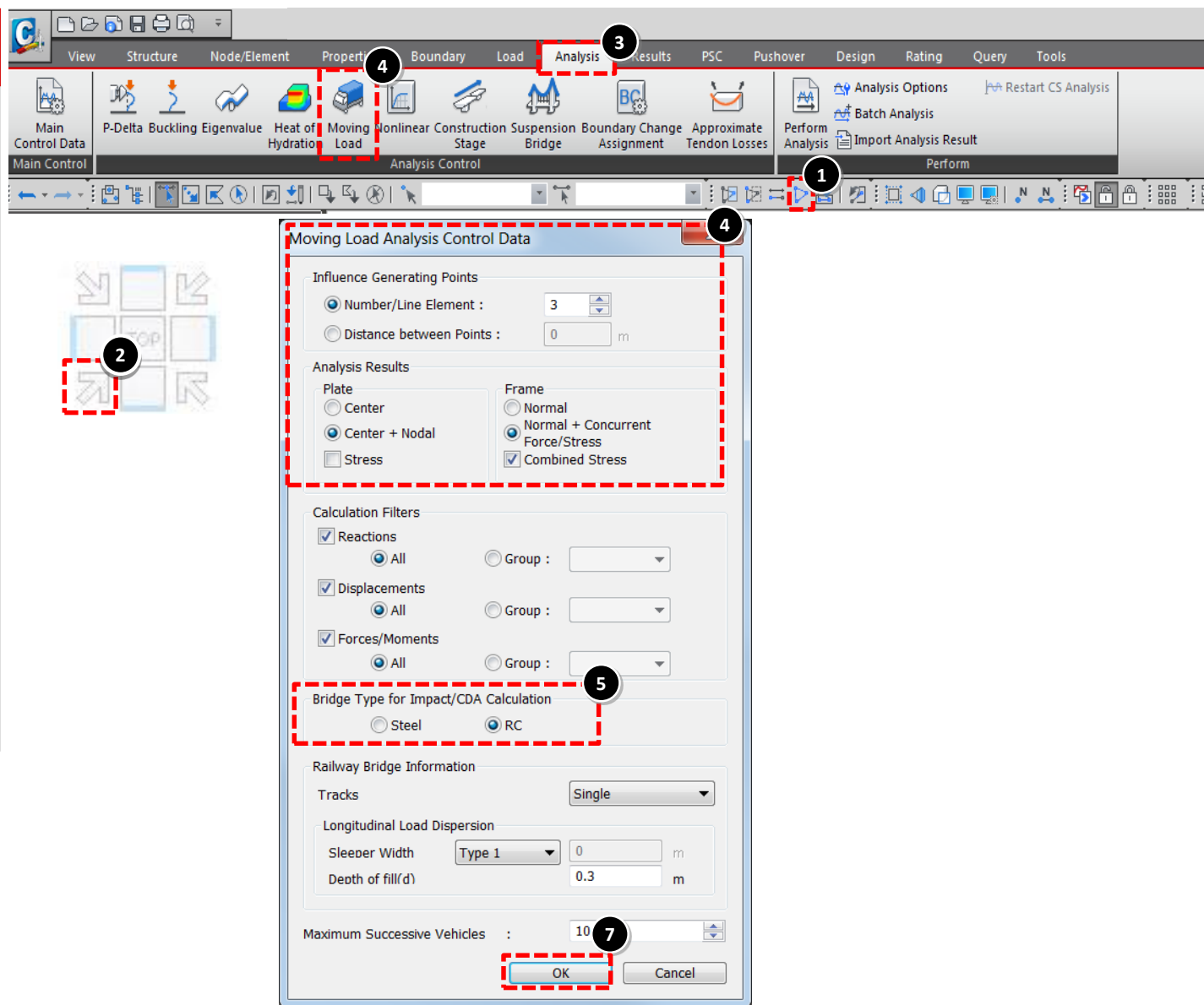
Sub-Domain for Mid walls

- 1 Go to Node/Elements > Define Sub-Domain
- 2 Domain Name > **Side walls**
 Sub Domain Name > **Mid walls**
 Member Type > **Plate Column (1D)**
 Rebar Direction > **Local**
- 3 Select Mid wall Elements as in image
- 4 Click on > **Add**



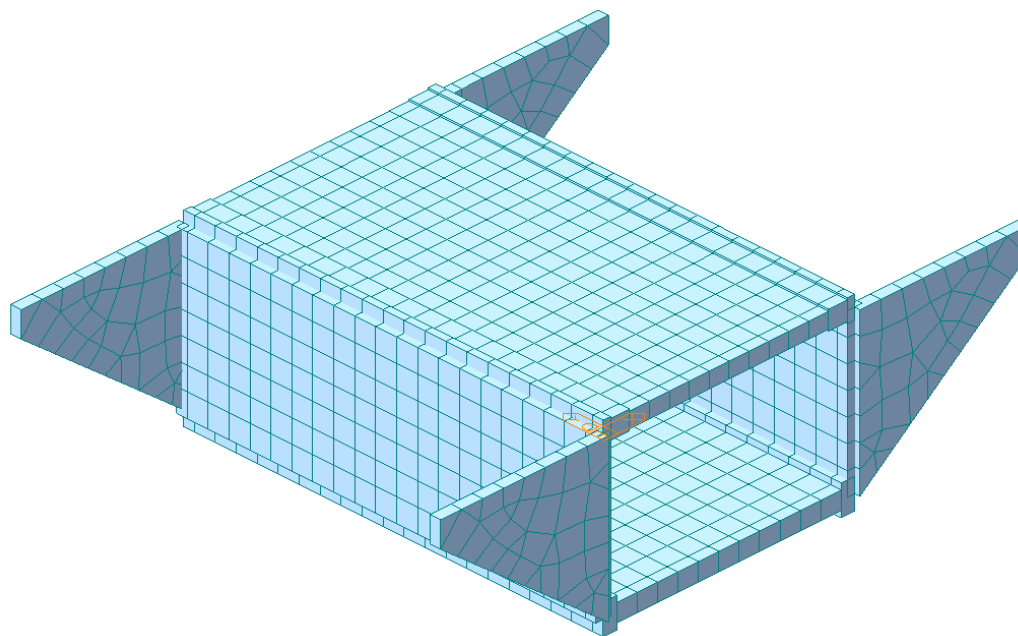
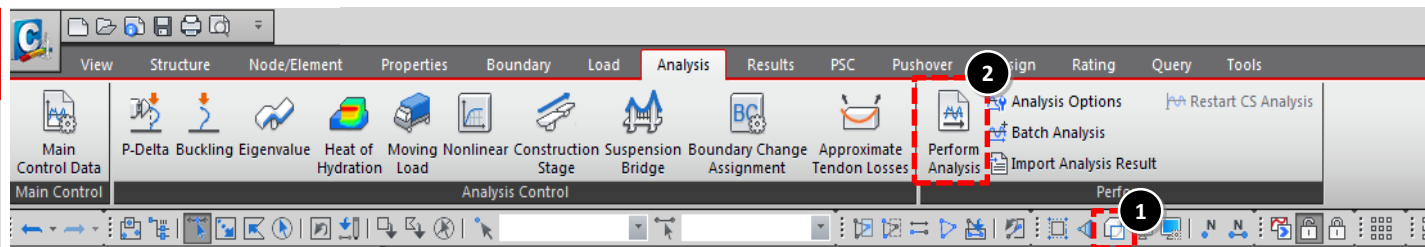
Moving Load Analysis Control

- 1 Click on > “Activate All”
- 2 Go to Iso View
- 3 Go to “Analysis” tab
- 4 Click “Moving Load”
- 5 In moving Load Analysis Control Data Window:
 - Enter Number/Line Elements > 3
 - Under Analysis Results > Frame
 - Select “Normal + Concurrent Force/Stress”
 - Check “Combined Stress Calculation”
- 6 Select Bridge Type for Impact
- Calculation > “RC”
- 7 Click “OK”



Perform Analysis

- 1 Click on > Hidden
- 2 Click “Perform Analysis” to Run Analysis



Auto Generation of LCB

- 1 Go to “Results” tab
- 2 Click “Load Combinations”
- 3 Go to > Concrete Design
- 4 Click on > Auto Generation
- 5 Select code > IRC6:LSD
- 6 Check off > Select All
- 7 Check on > Thermal Act.

The screenshot illustrates the steps for Auto Generation of Load Combinations (LCB) in the software. The interface shows the 'Results' tab selected in the top menu bar. The 'Load Combinations' dialog box is open, showing a list of load combinations. The 'Automatic Generation of Load Combinations' dialog box is also open, showing options for code selection, manipulation of construction stage load case, and leading variables.

Load Combinations Dialog Box:

No	Name	Active	Type	E	Description
1	LCB1	Stren	Add	<input type="checkbox"/>	No. I
2	LCB2	Stren	Add	<input type="checkbox"/>	No. IIA
3	LCB3	Stren	Add	<input type="checkbox"/>	No. IIB
4	LCB4	Stren	Add	<input type="checkbox"/>	No. IIB
5	LCB5	Stren	Add	<input type="checkbox"/>	No. V
*				<input type="checkbox"/>	

Automatic Generation of Load Combinations Dialog Box:

Option: ☒ Add ☐ Replace

Code Selection: ☐ Steel ☒ Concrete ☐ SRC ☐ Steel Composite

Design Code: **IRC:6 LSD**

Manipulation of Construction Stage Load Case: ☒ ST Only ☐ CS Only ☐ ST+CS
ST : Static Load Case CS : Construction Stage

Leading Variables: ☒ Select All ☐ Wind Loads ☒ Thermal Act. ☐ Snow Loads ☐ Construction Loads ☒ Traffic Loads ☒ Road Traffic

Buttons: OK, Cancel

Reaction

- 1 Go to “Results” tab
- 2 Click “Reactions” > “Reaction Forces/Moments”
- 3 Select “Load Cases/Combinations” > ST:Dead Load
- Select “Components” > Fxyz
- Select “Type of Display” > “Values”
- Click “Apply”
- 4 To View results in Tabular format,
Go to “Result” > “Results Tables”>
- 5 “Reactions”
- Select “Load Cases/Combination”
- 6 Click “OK”
- Check the values in new window
“Result-[Reaction]”

Reaction Forces/Moments

Load Cases/Combinations: ST: Dead Load

Step:

Components: ☐ FX ☐ FY ☐ FZ ☒ FXYZ ☐ MX ☐ MY ☐ MZ ☐ MXYZ ☐ Mb ☐ Local (if defined)

Type of Display: ☒ Values ☐ Legend ☐ Arrow Scale Factor: 1.000000

Records Activation Dialog

Node or Element: All None Inverse Prev

Node: Ito372

Select Type: Element Type Add Delete Replace Intersect

Loadcase/Combination: Dead Load(ST) Temp. Gradient(+)(ST) Temp. Gradient(-)(ST) EP w/o Water(ST) Shrinkage Load(ST) Class A(MV:all) Class A(MV:max) Class A(MV:min) Class 70R(MV:all) Class 70R(MV:max) Class 70R(MV:min) CLCB1(CBC:all) CLCB1(CBC:max)

OK Cancel

Result-[Reaction]

Node	Load	FX (kN)	FY (kN)	FZ (kN)	MX (kN*m)	MY (kN*m)	MZ (kN*m)
350	Dead L	0.000000	0.000000	21.561548	0.000000	0.000000	0.000000
351	Dead L	0.000000	0.000000	29.485002	0.000000	0.000000	0.000000
352	Dead L	0.000000	0.000000	29.486849	0.000000	0.000000	0.000000
353	Dead L	0.000000	0.000000	29.486999	0.000000	0.000000	0.000000
354	Dead L	0.000000	0.000000	29.486836	0.000000	0.000000	0.000000
355	Dead L	0.000000	0.000000	29.486836	0.000000	0.000000	0.000000
356	Dead L	0.000000	0.000000	29.486999	0.000000	0.000000	0.000000
357	Dead L	0.000000	0.000000	29.486849	0.000000	0.000000	0.000000
358	Dead L	0.000000	0.000000	29.485002	0.000000	0.000000	0.000000
359	Dead L	0.000000	0.000000	21.561548	0.000000	0.000000	0.000000
360	Dead L	0.000000	0.000000	6.820938	0.000000	0.000000	0.000000
361	Dead L	0.000000	0.000000	4.485243	0.000000	0.000000	0.000000
362	Dead L	0.000000	0.000000	14.175826	0.000000	0.000000	0.000000
363	Dead L	0.000000	0.000000	19.381752	0.000000	0.000000	0.000000
364	Dead L	0.000000	0.000000	19.381960	0.000000	0.000000	0.000000
365	Dead L	0.000000	0.000000	19.381993	0.000000	0.000000	0.000000
366	Dead L	0.000000	0.000000	19.381990	0.000000	0.000000	0.000000
367	Dead L	0.000000	0.000000	19.381990	0.000000	0.000000	0.000000
368	Dead L	0.000000	0.000000	19.381993	0.000000	0.000000	0.000000
369	Dead L	0.000000	0.000000	19.381960	0.000000	0.000000	0.000000
370	Dead L	0.000000	0.000000	19.381752	0.000000	0.000000	0.000000
371	Dead L	0.000000	0.000000	14.175826	0.000000	0.000000	0.000000
372	Dead L	0.000000	0.000000	4.485243	0.000000	0.000000	0.000000
SUMMATION OF REACTION FORCES PRINTOUT							
Load	FX (kN)	FY (kN)	FZ (kN)				
Dead L	0.000000	0.000000	1566.315625				

Displacement Contour

- 1 Go to “Results” tab
- 2 Click “Deformations” > “Displacement Contour”
- 3 Select “Load Cases/Combinations” > **ST:Dead Load**
 Select “Components”
 Click Type of Display > “Contour”
 and “Legend”
 Click “Apply”
- 4 See the Contour diagram in the “Model View” window

The screenshot shows the MIDAS/Civil POST-PROCESSOR interface. The 'Results' tab is selected. In the 'Reactions' section, 'Displacement Contour' is chosen. The 'Tree Menu' shows 'Displacement Contour' selected. The 'Load Cases/Combinations' dropdown is set to 'ST: Dead Load'. The 'Step' dropdown is set to '1'. The 'Components' section has 'DXYZ' selected. The 'Type of Display' section has 'Contour' and 'Legend' checked. The 'Apply' button is visible.

On the right, the 'DISPLACEMENT RESULTANT' table shows a color scale from 0.00 to 5.08. Below this, the 'ST: DEAD LOAD' table shows 'MAX : 60' and 'MIN : 277'. The 'VIEW-DIRECTION' is set to X: -0.483, Y: -0.837, Z: 0.259.

On the far right, a 3D model of a box culvert is shown with a displacement contour plot. The top surface is red, indicating the highest displacement, while the bottom and side surfaces are yellow and orange, indicating lower displacement.

Note: By Invoking “ ” the tables of any component of result and load cases can be checked.

Plate Forces/Moments

- 1 Go to “Results” tab
- 2 Click “Forces” > “Plate Forces/Moments”
- 3 Select “Load Cases/Combinations” > “ST:Dead Load”
Select “Components”
Click Type of Display > “Cutting Diagram”
- 4 Click on “Top View”
- 5 Select desired portion using “Select Single”
- 6 Click on “Activate”
- 7 Click on > Display Node Numbers
- 8 Click on “Iso view”

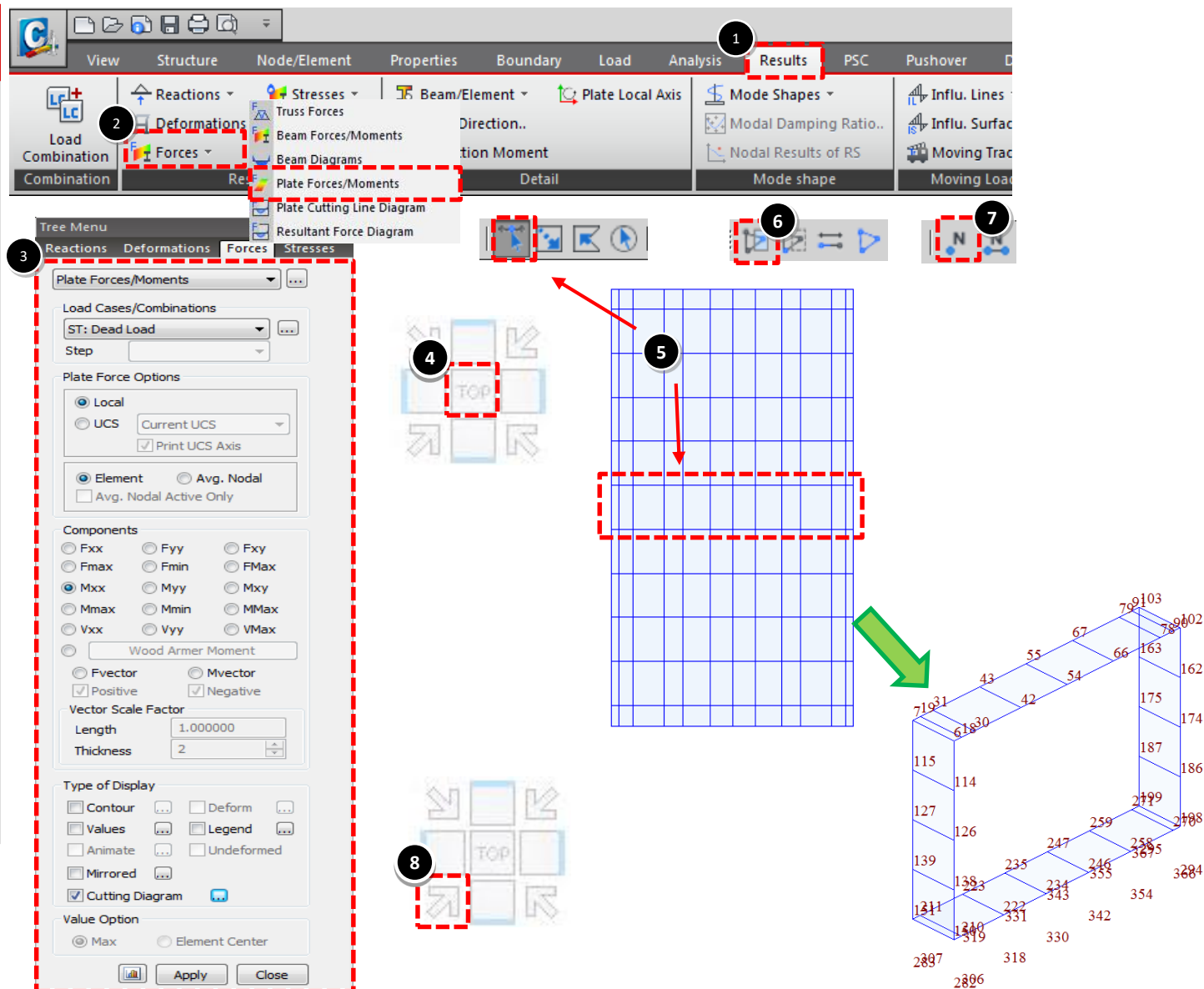



Plate Cutting Diagram

- Click on  and go to **Cutting Diagram** Tab

Enter Name: **Top**

Click in **Pnt1** text box

Click node **6** then **102** in model

Click on **Add**

Enter Name: **Right**

Click in **Pnt1**: text box

Click node **102** then **198** in model

Click on **Add**

Enter Name: **Bottom**

Click in **Pnt1**: text box

Click node **150** then **198** model

Click on **Add**

Enter Name: **Left**

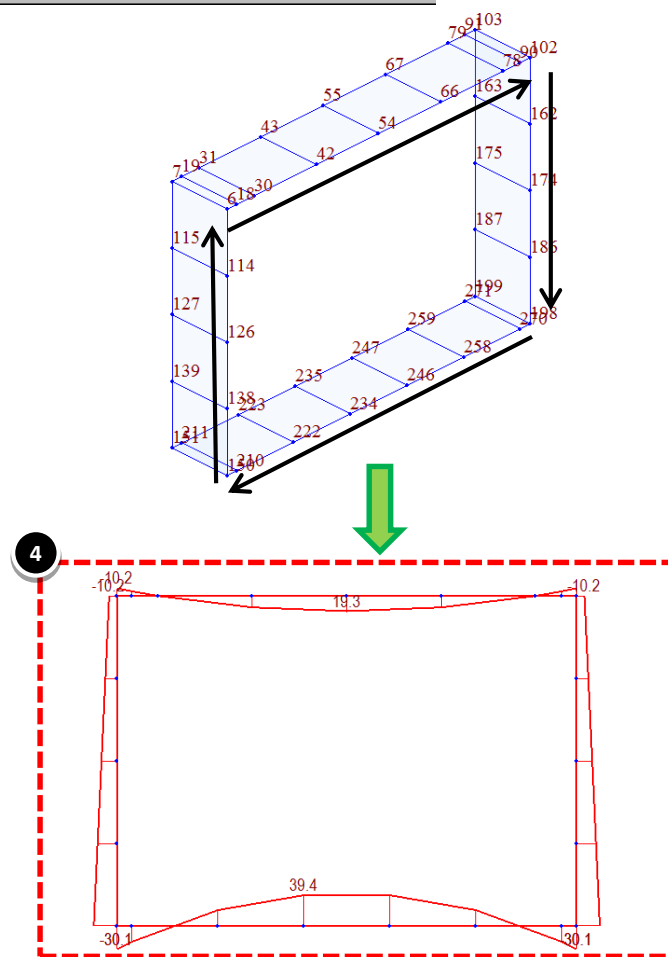
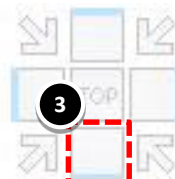
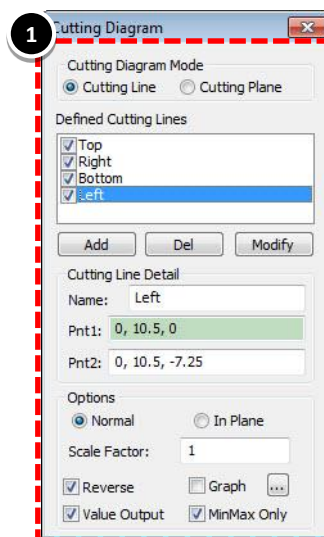
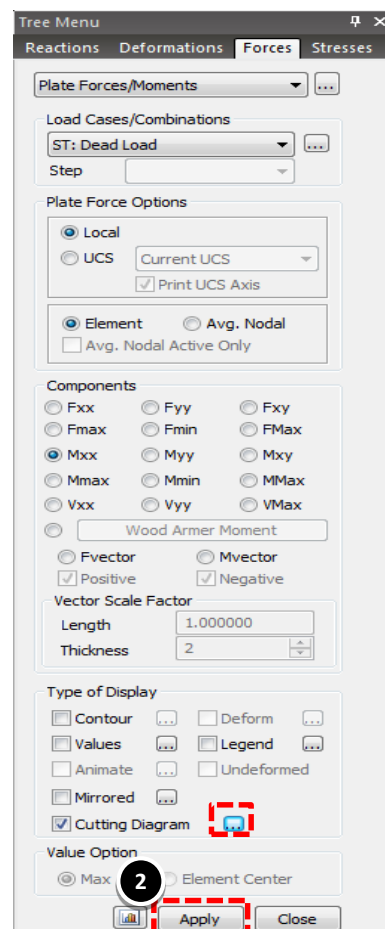
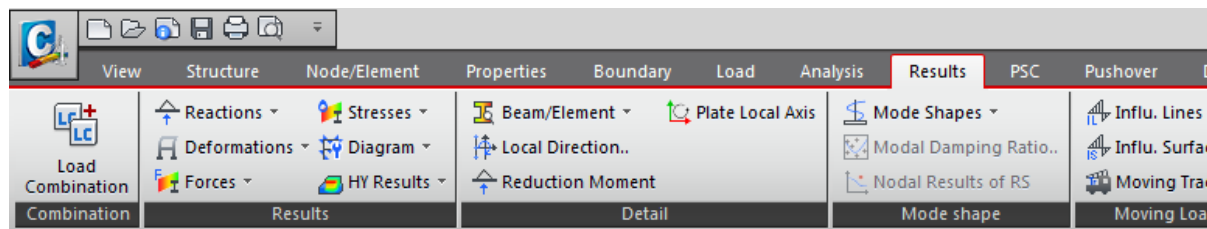
Click in **Pnt1**: text box

Click node **150** then **6** in model

Click on **Add**

- Click > **“Reverse”**
- Click > **“Apply”**
- Click on **“Front View”**

See the cutting diagram in the **“Model View”** window



Moving Load Tracer

- 1 Click on “Activate All”
- 2 Click on “Iso view”
- 3 Click on >“Display node numbers”
- 4 Go to “Results” tab
- 5 Click “Moving Load Tracer” > “Plate Forces/Moments...”
- 6 Go to Tree Menu

Select Moving load case > “MVmax :

Class A”

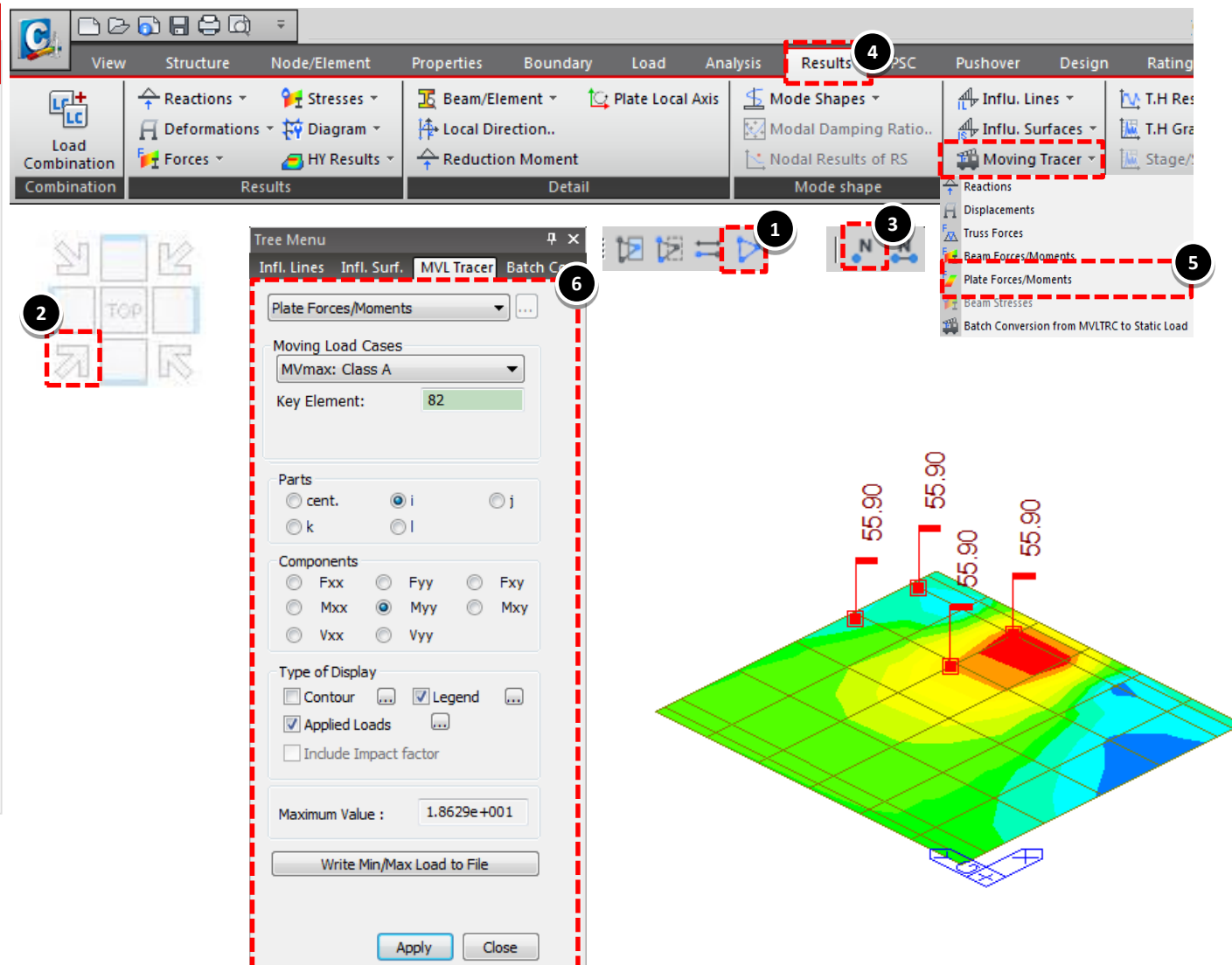
Select Key Element > “82”

Select Part > “i”

Select Component > “Myy”

Click > “Apply”

See in “Model View” window the
Vehicle position



Modify Concrete Material

- 1 Go to “Design” tab
 - 2 Select “IRC 112-2011”
 - 3 Click “RC Design”
 - 4 Select “Modify Concrete Material”
 - 5 Click on Material Name “M35”
- Under Concrete Material Selection
- Select Code: **IS(RC)**
- Select Grade: **M35**
- Under Rebar Selection
- Select Code: **IS(RC)**
- Select Grade of Main Rebar: **Fe500**
- Select Grade of Sub-Rebar: **Fe415**
- 6 Click **Modify & Close**

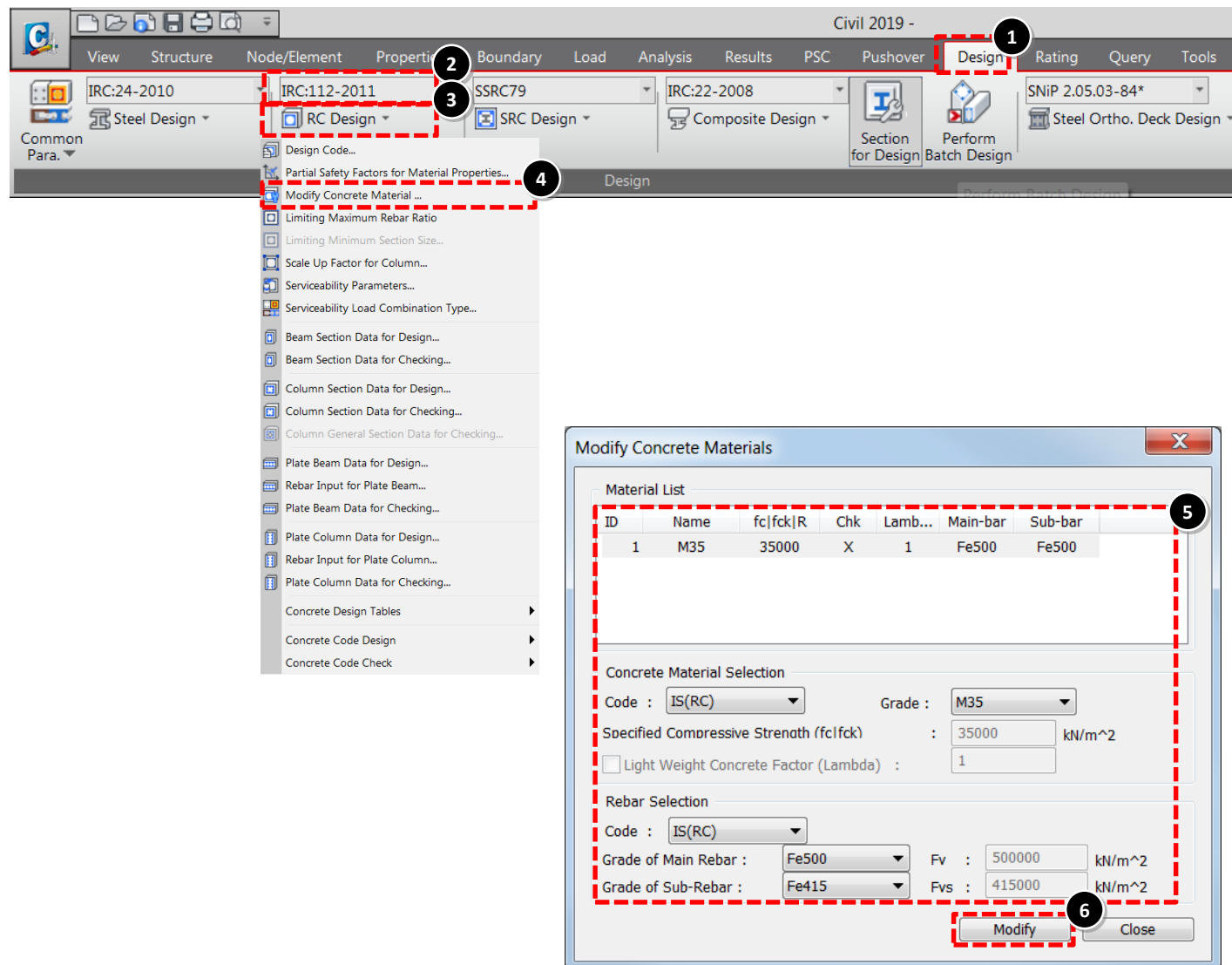


Plate Beam Data for Design

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select “Plate Beam Data for Design”
- 5 Sub Domain Name > **Left End Slabs**
 Plate Force Option > **Avg. Nodal**
 Main Rebar Direction > **Dir. 1**
 Dt > **0.045 m**
 Db > **0.045 m**
 Click on > **Add**
- 6 Click on > **Close**
- 7 Repeat **Step 5** to add Right End and Mid Slabs

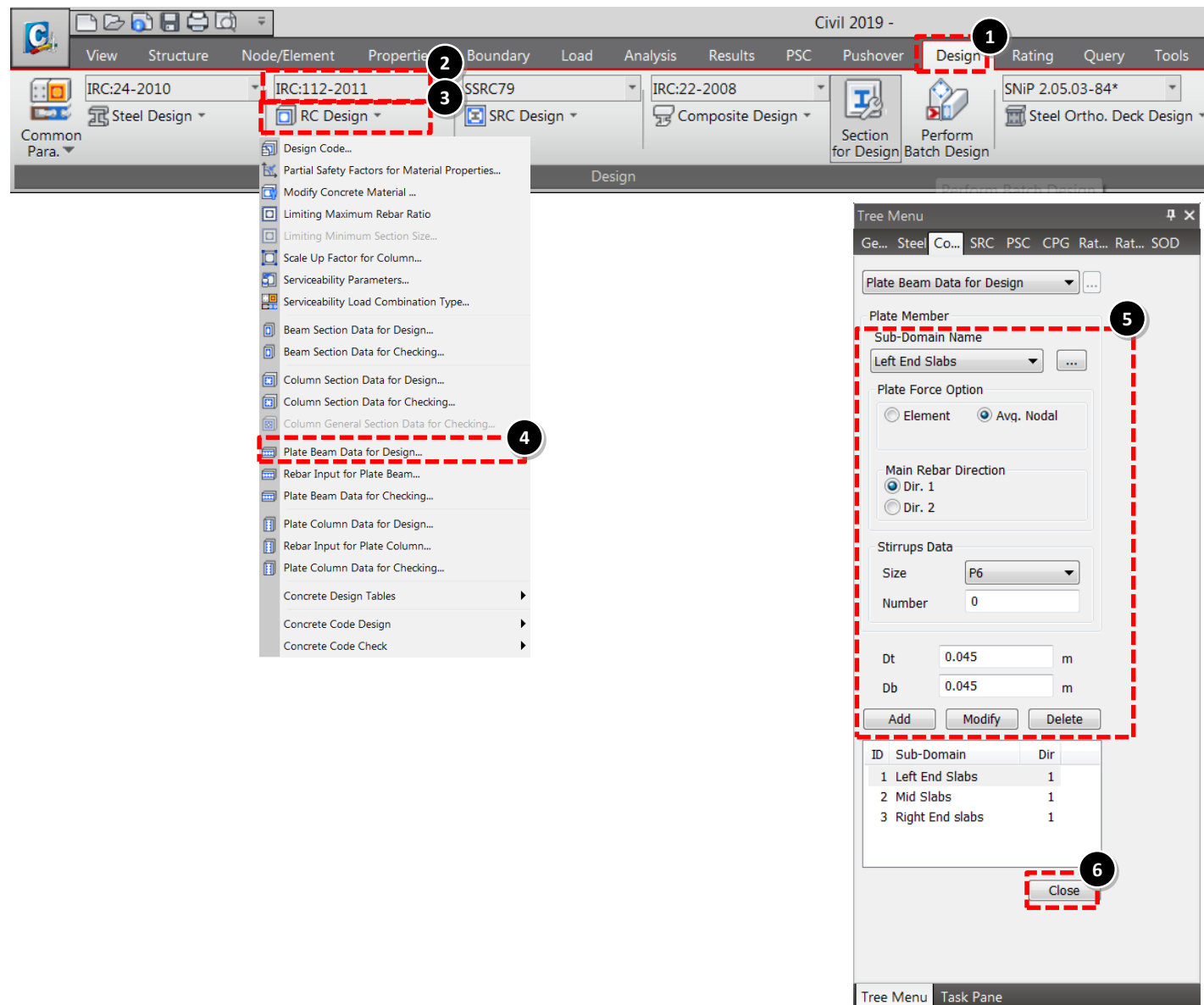


Plate Column Data for Design

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select “Plate Column Data for Design”
- 5 Sub Domain Name > **Bottom Walls**
 Plate Force Option > **Avg. Nodal**
 Main Rebar Direction > **Dir. 1**
 Dt > **0.045 m**
 Db > **0.045 m**
 Click on > **Add**
- 6 Click on > **Close**
- 7 Repeat **Step 5** to add Right End and Mid Slabs

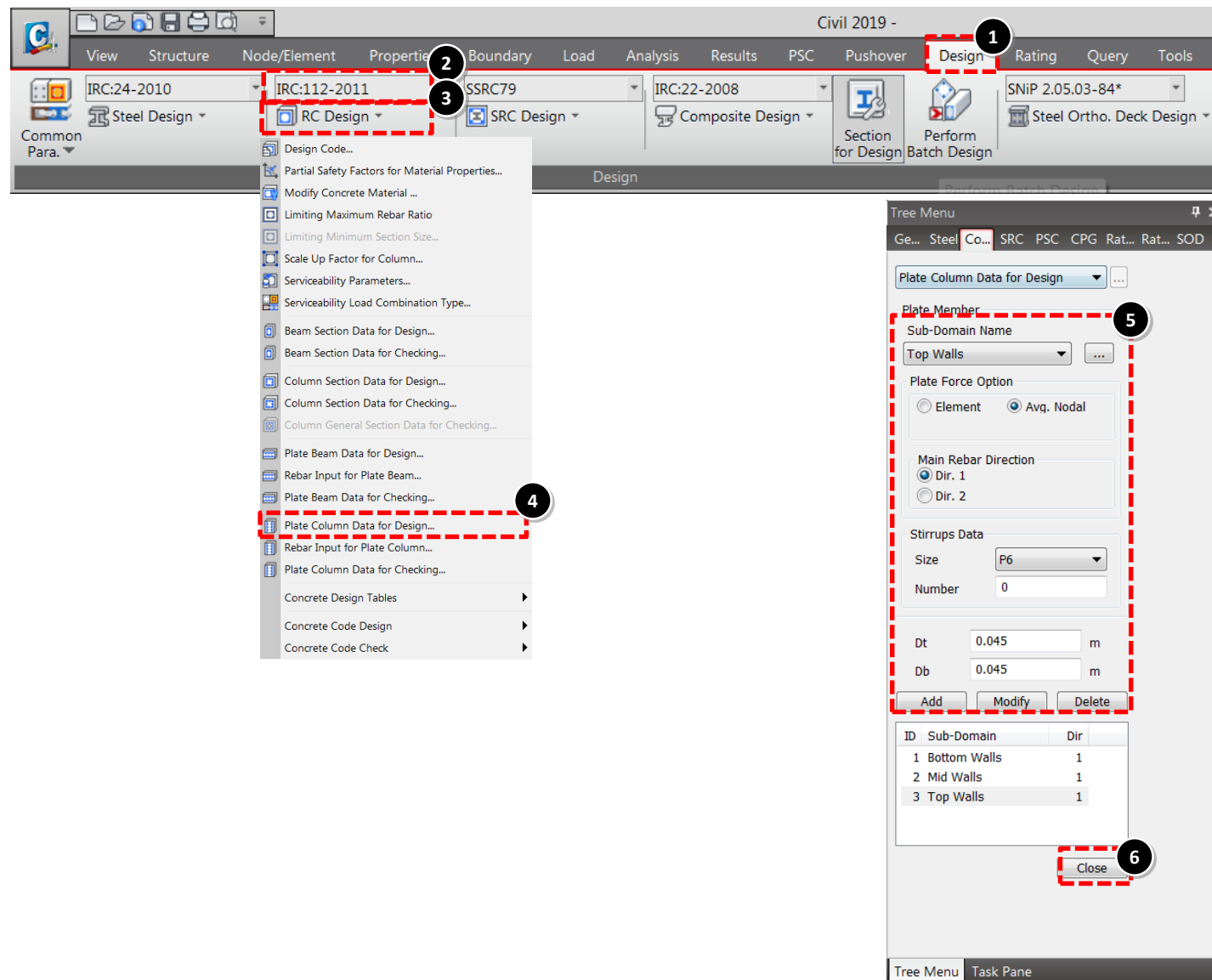


Plate Beam Design

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select Concrete Code Design > Plate Beam Design
- 5 Select The Left End sub-Domain
- 6 Click on > Graphic

Design

Code: IRC:112-2011 Unit: kN.m / m

Sub-Domain	SEL	Major Dir	CHK	Stirrup
Left End	<input type="checkbox"/>	Dir1	OK	No Stirrup
Mid Slab	<input type="checkbox"/>	Dir1	OK	No Stirrup
Right End	<input type="checkbox"/>	Dir1	OK	No Stirrup

2. Section Diagram

Thickness : 0.325 m

Element No : 23

Rebar Pattern

Top Required Rebar Area = 0.00106729 m²/m

Bot Required Rebar Area = 0.000724552 m²/m

Required Stirrups Spacing : No Stirrup

3. Bending Capacity

	Top(Negative)	Bottom(Positive)
Mu	130.38	82.87
Element No.	177	23
Load Combination	c.CB1	c.CB1
Mr	134.30	84.64
Check Ratio (Mu/Mr)	0.9708	0.9791

4. Shear Capacity

	Value	Check
Element No.	6	
Load Combination	c.CB2	
Applied Shear Force	V_Ed = 264.039	
Shear Strength (Out of plane)	V_Rd = 272.020	
Required Stirrups Spacing	0.00000 m	
Shear Ratio	V_Ed/V_Rd = 264.039 / 272.020	= 0.971 < 1.000 O.K

Plate Column Design

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select Concrete Code Design > Plate Beam Design
- 5 Select The Bottom walls sub-Domain
- 6 Click on > Detail

Design

Design Code...
Partial Safety Factors for Material Properties...
Modify Concrete Material ...
Limiting Maximum Rebar Ratio
Limiting Minimum Section Size...
Scale Up Factor for Column...
Serviceability Parameters...
Serviceability Load Combination Type...
Beam Section Data for Design...
Beam Section Data for Checking...
Column Section Data for Design...
Column Section Data for Checking...
Column General Section Data for Checking...
Plate Beam Data for Design...
Rebar Input for Plate Beam...
Plate Beam Data for Checking...
Plate Column Data for Design...
Rebar Input for Plate Column...
Plate Column Data for Checking...
Concrete Design Tables
Concrete Code Design
Concrete Code Check

Plate Column Design Result Dialog

Code : IRC:112-2011 Unit : KN, m / m

Sub-Domain	SEL	Major Dir	CHK	Ratio_V	Asw
Bottom	<input checked="" type="checkbox"/>	Dir1	OK	0.3737	0.0000
Mid Wall	<input type="checkbox"/>	Dir1	OK	0.1871	0.0000
Top Wall	<input type="checkbox"/>	Dir1	OK	0.3752	0.0000

MIDAS/Text Editor - [3D Culvert_After Wizard]

MIDAS/Civil - RC-Plate Column Design

```

*.MIDAS/Civil - RC- PLATE COLUMN DESIGN
*.PROJECT :
*.DESIGN CODE : IRC:112-2011,
*.SUB-DOMAIN : Bottom Walls Member Type = PLATE COLUMN(1D), Dir = 1 )

*.DESCRIPTION OF PLATE COLUMN DATA : (Elem : 122)
Thickness = 0.350 m.
Unit Width = 1 m.
Concrete Strength (fck) = 35000.000 KPa.
Main Rebar Strength (fy) = 500000.000 KPa.
Stirrups Strength (fyw) = 415000.000 KPa.
Modulus of Elasticity (Es) = 200000000.000 KPa.

< Selected Elements >
122to132 166to176
< Positive Bending Moment and Axial Forces >
P-M_Ed = 0.00 kN-m/m., ELEM = 122, LCB = 1+, NODE = Center
P-N_Ed = 105.42 kN/m.
< Negative Bending Moment and Axial Forces >
N-M_Ed = 128.79 kN-m/m., ELEM = 122, LCB = 1-, NODE = 145
N-N_Ed = 314.14 kN/m.
< Shear Force >
V_Ed = 53.45 kN/m. , ELEM = 122, LCB = 1+, NODE = Center

*.REINFORCEMENT PATTERN :
Dt = 0.045 ( m. )
Db = 0.045 ( m. )

Stirrups : No BarNum

=====
[[[*]]] ANALYZE AXIAL-MOMENT RESISTANCE OF POSITIVE MOMENT REGION.
  
```

End Slabs

- 1 Go to “Design” tab
 - 2 Select “IRC 112-2011”
 - 3 Click “RC Design”
 - 4 Select “Rebar Input for Beam”
 - 5 Name > End Slabs
- Under Main Rebar > CTC > Top
- CTC > 0.2; Size 1 > P20; Size 2 > P12
- Dt > 0.045
- Bottom >**
- CTC > 0.2; Size 1 > P16; Size 2 > P12
- Dt > 0.045
- 6 Under Distribution Rebar > CTC >

Top

CTC > 0.2; Size 1 > P10

Bottom >

CTC > 0.2; Size 1 > P10

- 7 Click on > Add

The screenshot shows the Civil 2019 software interface with the Design tab selected. The menu bar includes View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, PSC, Pushover, Design, Rating, Query, and Tools. The Design tab is active, and the RC Design option is highlighted. The Rebar Input for Plate Beam dialog box is open, showing the Main Rebar and Distribution Bar sections. The dialog box contains fields for Name, Main Rebar, Distribution Bar, and various design parameters. The Rebar Input for Plate Beam dialog box is shown twice, illustrating the steps for adding rebar.

Rebar Input for Plate Beam

Name: End Slabs

Main Rebar: ☐ Num ☒ CTC

Top

Layer	CTC	Size1	Size2	Dt
1	0.2	P20	P12	0.045

Bottom

Layer	CTC	Size1	Size2	Db
1	0.2	P16	P12	0.045

☐ Stirrup

Size: P6

Spacing: 0 m

Number: 0

Add Modify Delete

ID Name

Close

Mid Slabs

1 Name > **Mid Slabs**

Under Main Rebar > **CTC > Top**

CTC > 0.2; Size 1 > **P20**; Size 2 > **P12**

Dt > **0.045**

Bottom >

CTC > 0.2; Size 1 > **P16**; Size 2 > **P12**

Dt > **0.045**

2 Under Distribution Rebar > **CTC >**

Top

CTC > 0.2; Size 1 > **P10**

Bottom >

CTC > 0.2; Size 1 > **P10**

3 Click on > **Add**

Rebar Input for Plate Beam

Name: Mid Slabs

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.0010055 m²/m Layer: 1

Layer	CTC	Size1	Size2	Dt
1	0.2	P16	P12	0.045

Bottom

As: 0.0010055 m²/m Layer: 1

Layer	CTC	Size1	Size2	Db
1	0.2	P16	P12	0.045

☐ Stirrup

Size: P6

Spacing: 0 m

Number: 0

Add Modify Delete

ID	Name
1	End Slabs

Close

Rebar Input for Plate Beam

Name: Mid Slabs

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.0003925 m²/m Layer: 1

Layer	CTC	Size1
1	0.2	P10

Bottom

As: 0.0003925 m²/m Layer: 1

Layer	CTC	Size1
1	0.2	P10

☐ Stirrup

Size: P6

Spacing: 0 m

Number: 0

Add Modify Delete

ID	Name
1	End Slabs

Close

Top and Bottom Walls

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select “Rebar Input for Column”
- 5 Name > **Top and Bottom Walls**

Under Main Rebar > CTC > Top

CTC > 0.2; Size 1 > P16; Dt > 0.045

Bottom >

CTC > 0.2; Size 2 > P12

Dt > 0.045

- 6 Under Distribution Rebar > CTC >

Top

CTC > 0.2; Size 1 > **P10**

Bottom >

CTC > 0.2; Size 1 > **P10**

- 7 Click on > **Add**

The screenshot shows the Civil 2019 software interface. The 'Design' tab is selected in the top menu bar. The 'RC Design' menu is open, showing various design options. The 'Rebar Input for Plate Column' dialog box is shown twice, illustrating the input for the top and bottom walls. The dialog box contains fields for Name, Main Rebar, Distribution Bar, and tables for Top and Bottom rebar input. The 'Add' button is highlighted in the bottom right corner of the dialog box.

Rebar Input for Plate Column (Top Wall)

Layer	CTC	Size1	Size2	Dt
1	0.2	P16		0.045

Rebar Input for Plate Column (Bottom Wall)

Layer	CTC	Size1	Size2	Db
1	0.2	P12		0.045

Rebar Input for Plate Column (Top Wall)

Layer	CTC	Size1
1	0.2	P10

Rebar Input for Plate Column (Bottom Wall)

Layer	CTC	Size1
1	0.2	P10

Mid Walls

1 Name > **Mid Walls**Under Main Rebar > **CTC > Top**CTC > 0.2; Size 1 > **P12**; Dt > **0.045****Bottom >**CTC > 0.2; Size 1 > **P12**; Dt > **0.045**2 Under Distribution Rebar > **CTC >****Top**CTC > 0.2; Size 1 > **P10****Bottom >**CTC > 0.2; Size 1 > **P10**3 Click on > **Add**

Rebar Input for Plate Beam

Name: Mid Slabs

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.0010055 m²/m Layer: 1

Layer	CTC	Size1	Size2	Dt
1	0.2	P16	P12	0.045

Bottom

As: 0.0010055 m²/m Layer: 1

Layer	CTC	Size1	Size2	Db
1	0.2	P16	P12	0.045

☐ Stirrup

Size: P6

Spacing: 0 m

Number: 0

Add Modify Delete

ID	Name
1	End Slabs

Close

Rebar Input for Plate Beam

Name: Mid Slabs

Main Rebar: Distribution Bar

☐ Num ☒ CTC

Top

As: 0.0003925 m²/m Layer: 1

Layer	CTC	Size1
1	0.2	P10

Bottom

As: 0.0003925 m²/m Layer: 1

Layer	CTC	Size1
1	0.2	P10

☐ Stirrup

Size: P6

Spacing: 0 m

Number: 0

Add Modify Delete

ID	Name
1	End Slabs

Close

Plate Beam Checking

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select Concrete Code Checking >
Plate Beam Design
- 5 Select The Left End sub-Domain
- 6 Click on > Graphic

1 Go to “Design” tab

2 Select “IRC 112-2011”

3 Click “RC Design”

4 Select Concrete Code Checking >
Plate Beam Design

5 Select The Left End sub-Domain

6 Click on > Graphic

2. Section Diagram

Element No : 23

Rebar Pattern

	Top(Negative)	Bottom(Positive)
Layer 1	P20,12@0.20	P16,12@0.20

Total Rebar Area Ast = 0.0025765 m²/2m
Using Stirrups Spacing : No Stirrup

3. Bending Moment Capacity

	Top(Negative)	Bottom(Positive)
Element No.	6	6
Load Combination	dLCB2	dLCB2
Applied Shear Force	V_Ed = 264.039	V_Ed = 264.039
Shear Strength (Out of plane)	V_Rd = 484.726	V_Rd = 484.726
Shear Ratio	V_Ed/V_Rd = 264.039 / 484.726	V_Ed/V_Rd = 264.039 / 484.726

Check Ratio (Mu/Mr) 0.6763 0.7152
Using Rebar(As) 0.0016 0.0010

4. Shear Capacity

Element No. 6
Load Combination dLCB2
Applied Shear Force V_Ed = 264.039
Shear Strength (Out of plane) V_Rd = 484.726
Shear Ratio V_Ed/V_Rd = 264.039 / 484.726 = 0.545 < 1.000 OK

5. Stress Check

	Concrete	Rebar
Element No.	177	177
(-) Load Combination	dLCB69	dLCB69

Plate Beam Check Result Dialog

Code : IRC:112-2011 Unit : kN. m / m
Results : ☒ Strength ☐ Serviceability

Sub-Domain	SEL	Major Dir	CHK
Left End Slabs	<input checked="" type="checkbox"/>	Dir1	OK
Mid Slabs	<input type="checkbox"/>	Dir1	OK
Right End slabs	<input type="checkbox"/>	Dir1	OK

☐ Connect Model View
Select All Unselect All Re-calculation
Detail Graphic
Files\Tutorial\Final Report ... >>>
Close

Plate Column Checking

- 1 Go to “Design” tab
- 2 Select “IRC 112-2011”
- 3 Click “RC Design”
- 4 Select Concrete Code Checking >
Plate Column Checking
- 5 Select The **Mid Walls** sub-Domain
- 6 Click on > **Detail**

The screenshot shows the Civil 19 software interface with the Design tab selected. The Design Code is set to IRC:112-2011, and the Design Type is RC Design. The Concrete Code Check menu is open, showing the Plate Column Checking option. The Plate Column Check Result Dialog is displayed, showing the Mid Walls sub-Domain selected for checking. The dialog also shows the results of the check, including the design code, unit, and the results of the strength and serviceability checks.

Design Tab:

- Design Code: IRC:112-2011
- Design Type: RC Design
- Design Code Check: Concrete Code Check

Plate Column Check Result Dialog:

Sub-Domain	SEL	Major Dir	CHK
Bottom Walls	<input type="checkbox"/>	Dir1	OK
Mid Walls	<input checked="" type="checkbox"/>	Dir1	OK
Top Walls	<input type="checkbox"/>	Dir1	OK

Reinforcement Pattern:

Location	i	di (m.)	Rebar	Asi (m^2/m.)
Top	1	0.012	F12@200.00	0.00057
Bottom	1	0.012	F12@200.00	0.00057

Concrete Code Check:

```

V_Ed = 0.00 kN/m. , ELEM = 100, LCB = 0.000000
*.REINFORCEMENT PATTERN :
Location i di ( m.) Rebar Asi ( m^2/m.)
Top 1 0.012 F12@200.00 0.00057
Bottom 1 0.012 F12@200.00 0.00057
Stirrups : No BarNum

=====
[[[*]]] ANALYZE AXIAL-MOMENT RESISTANCE OF POSITIVE MOMENT REGION.
=====

( ). Check area of tensile reinforcement (Rectangular-beam).
-. fyk = 500000.0000 KPa.
-. fctm = 0.259 * fck^(2/3) = 3209.9624 KPa.
-. As.min = MAX[ 0.26*(fctm/fyk)*bt*d, 0.0013*bt*d ] = 4.3952e-007 m^2.
-. As.max = 0.04 * (Bc*Hc) = 1.4000e-005 m^2.
-. As.prov = 5.6550e-007 m^2.
-. As.min < As.prov < As.max ----> O.K !

( ). Check moment resistance.
-. c = 0.1227 m.
-. Angle of N.A = 0.0000 deg.
-. Co + Cs = 1771.693 kN/m.
-. Ts = 245.870 kN/m.
-. N_max = 5945.724 kN/m.
-. N_Rd = 1525.824 kN/m.
-. M_Rd = 256.017 kN-m/m.
-. N_Ed/N_Rd = 0.037 ----> O.K !
-. M_Ed/M_Rd = 0.038 ----> O.K !
-. Ratio = 0.038 ----> O.K !
  
```