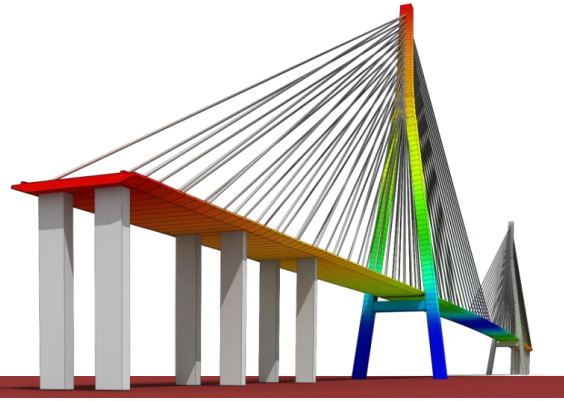
MIDAS *Technical Material*

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 + 2x 450	= 9.65 m

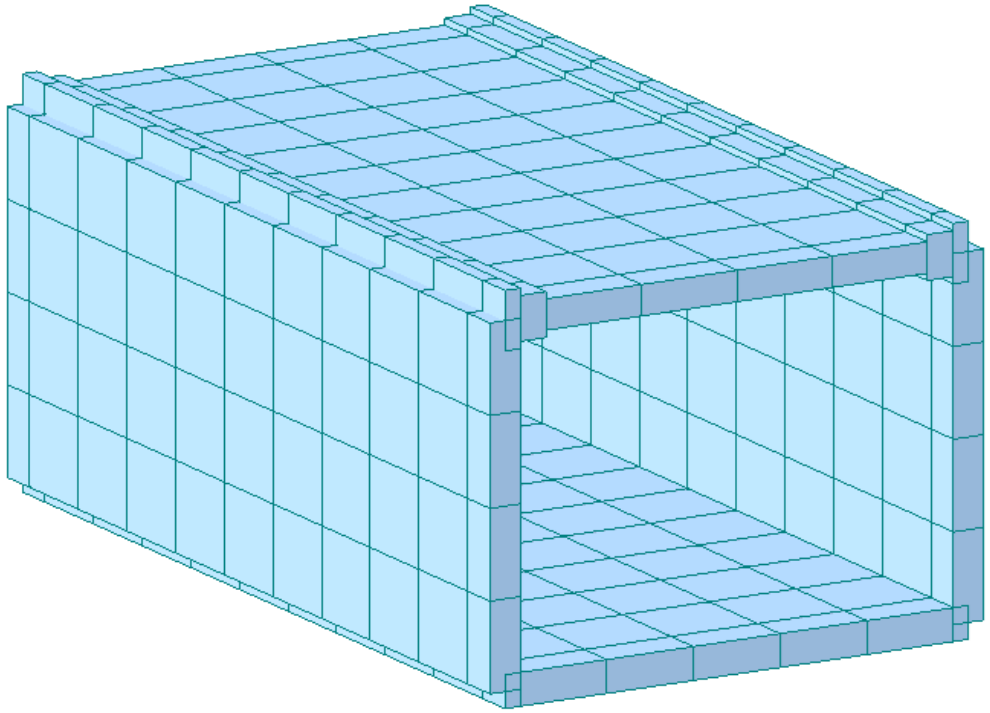


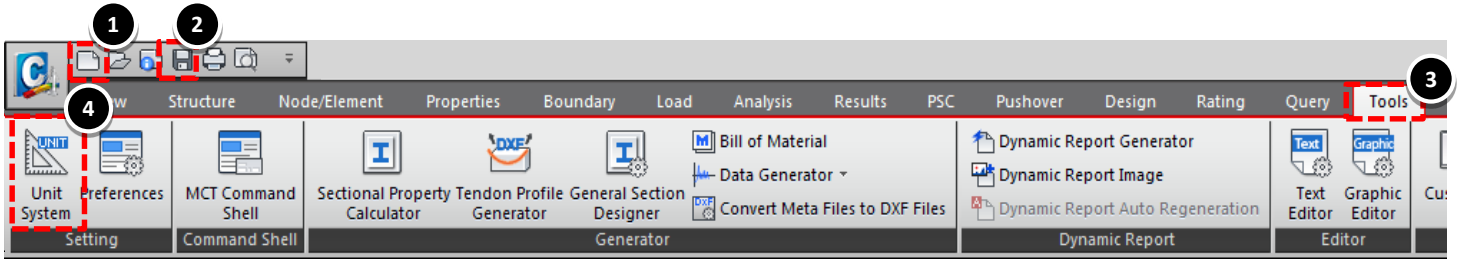
Table of Contents

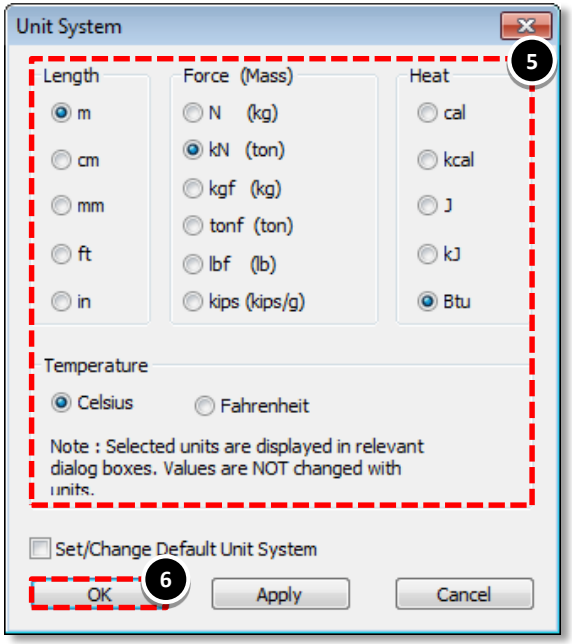
Initial Setting – Units Preference

Units Preference

- Invoke midas Civil
- Open New File
- Save as "**Box Culvert using wizard**"
- Go to > "**Tools**"
- Click on "**Unit System**"
- Select the unit system **[m, kN(ton), Celcius]**
- Click on **OK**

.

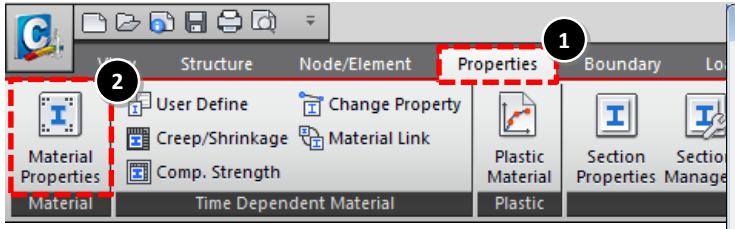


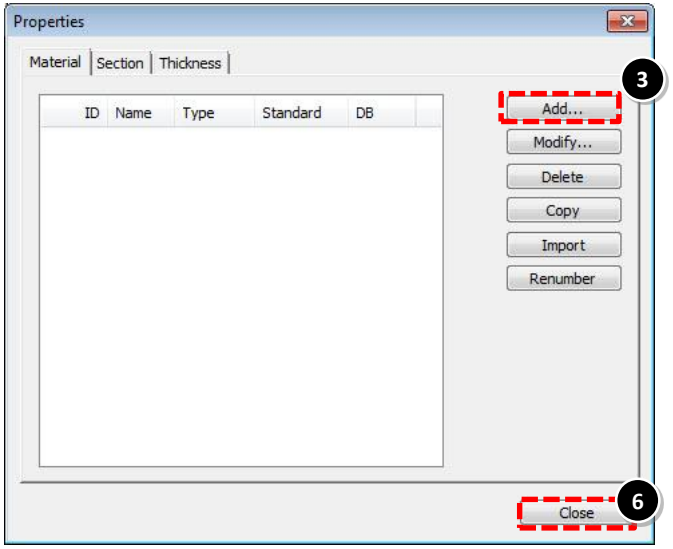


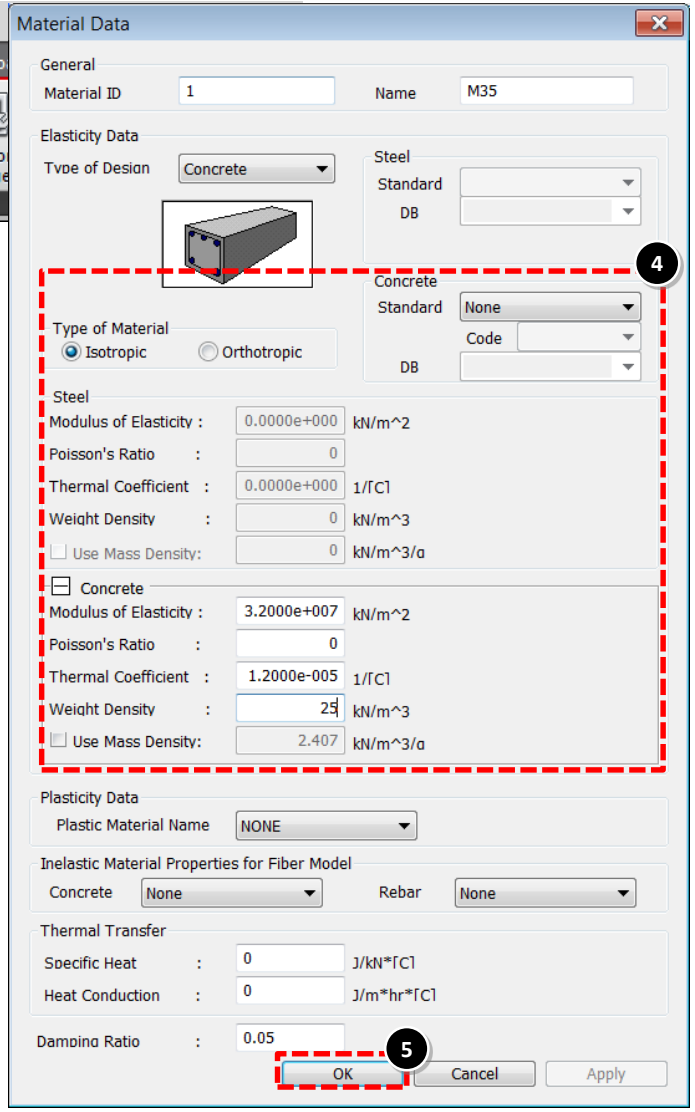
Model Generation – Definition of Properties

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 > **3.2e+007**
   - Poisson’s ratio > **0**
   - Thermal Coefficient > **1.2e-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

Step **01**

Model Generation – RC. Frame/Box Culvert Wizard

Longitudinal Tab

1. Go to “**Structure**” tab
2. Click on “**RC Frame/Box**”
3. Go to “**R.C.Frame/Box Culvert Wizard**” tab

   Go to “**Longitudinal**” tab
4. **Define data:**

   structure type > “**3-Dimensional**”

   Type of bridge > “**Box Culvert**”

   Material > “**M35**”

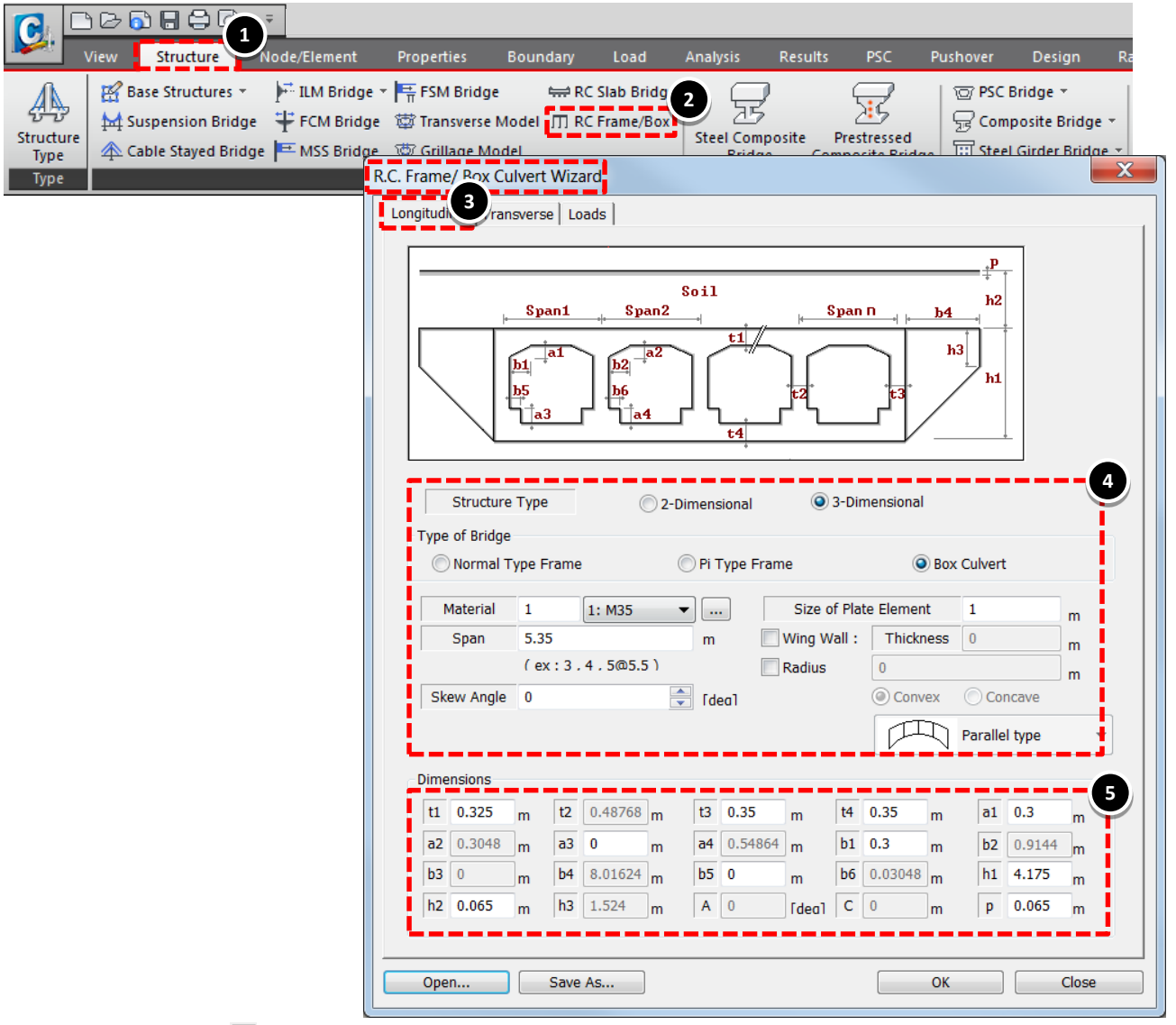
   Size of Plate Element > **1 m**

   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.

Model Generation – RC. Frame/Box Culvert Wizard

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**

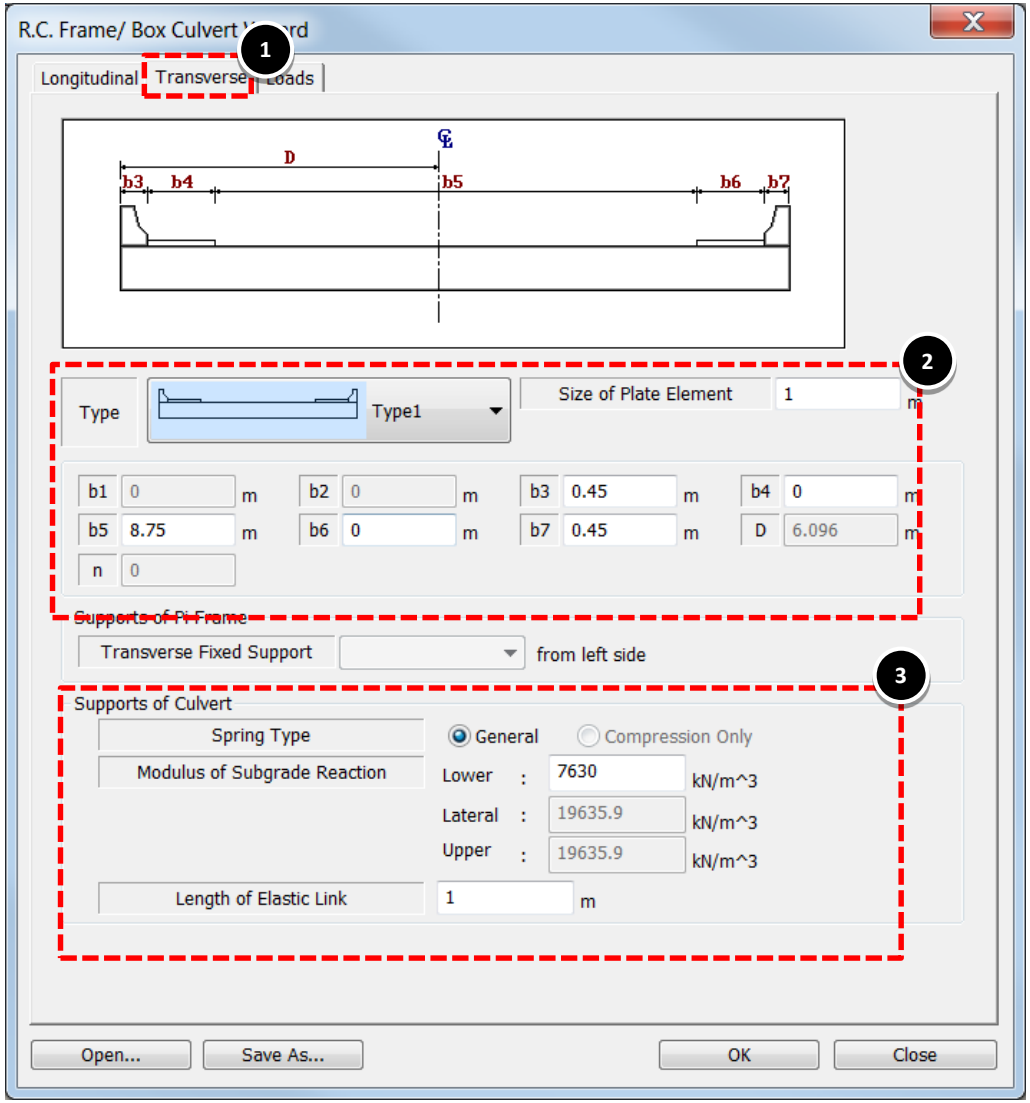
3. **Define data for supports of culvert:**

   Spring Type > "**General**"

   Modulus of Subgrade Reaction >

   Lower > **7630 kN/m³**

   Length of Elastic Link > **1 m**

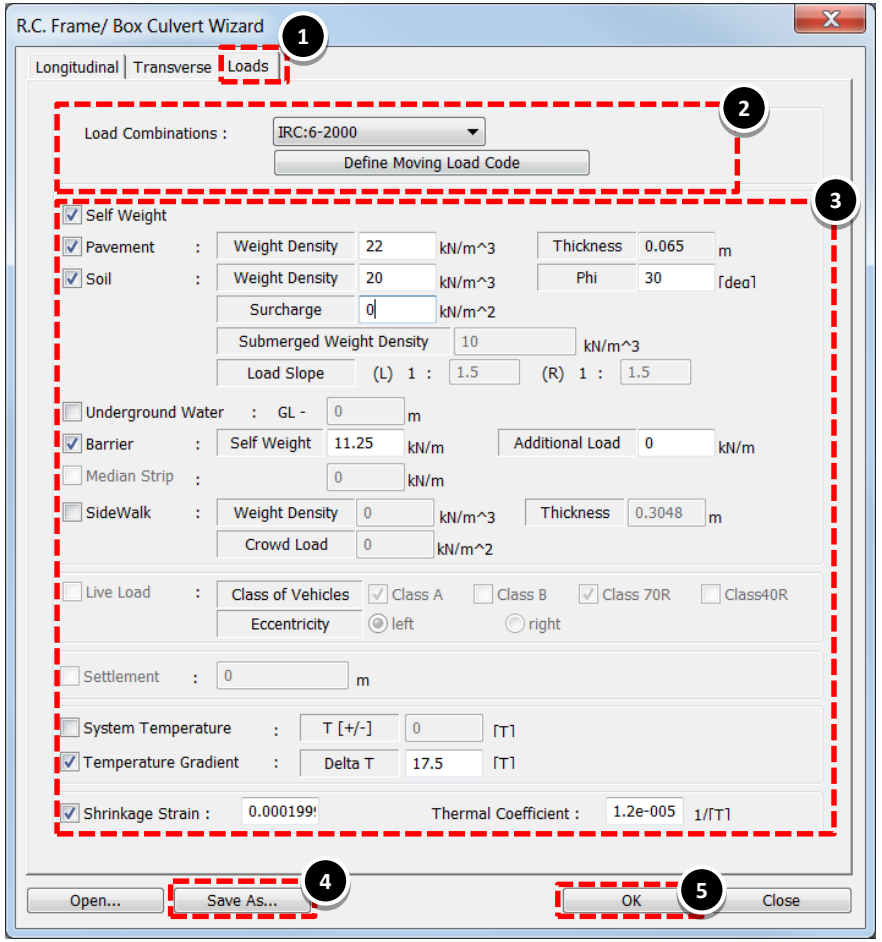


Step **01**

Model Generation – RC. Frame/Box Culvert Wizard

Load Tab

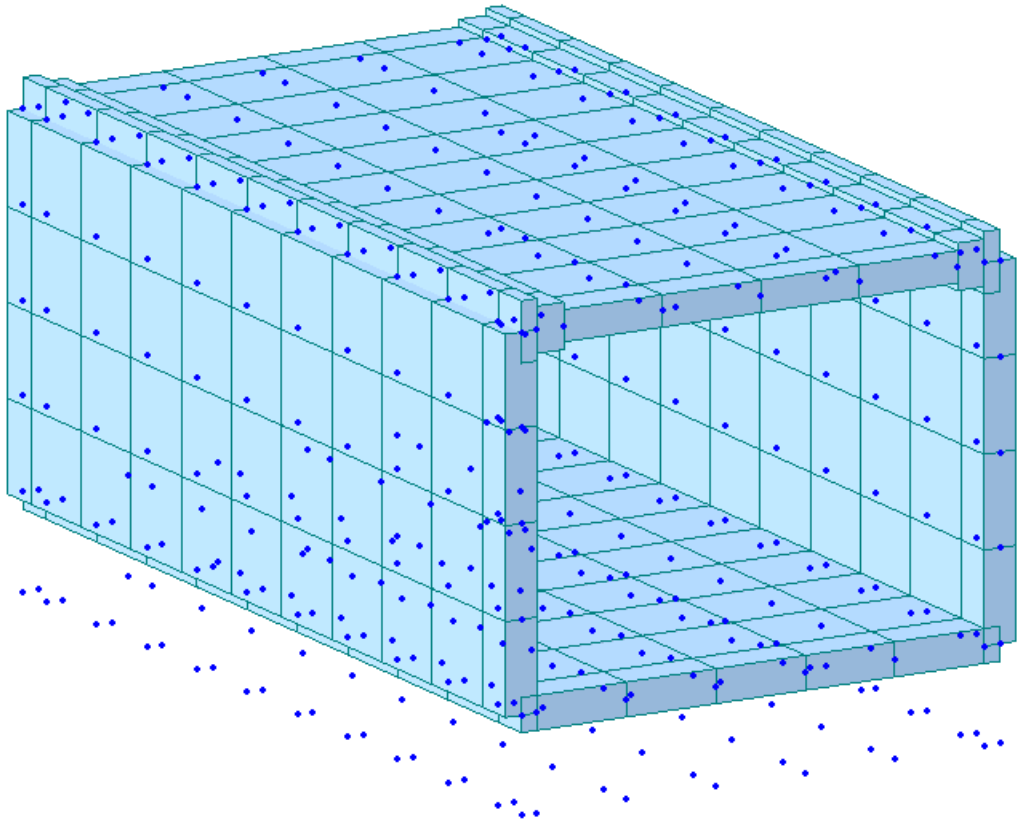
1. Go to "**Load**" tab
2. **Define Load Combination :**
   - Select code > "**IRC:6-2000**"
   - Click > **Define Moving Load Code**
   - Select Moving Load Code > **"India"**
3. Click > **"Self Weight"**
   - Click > **"Pavement"**
     - Weight Density > **22 kN/m²**
     - Thickness > **0.065m**
   - 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**



Note: Assuming area of crash barriers as 0.45 m^2 SW of CB = 0.45X 25 = 11.25 kN/m

Model Generation – RC. Frame/Box Culvert Wizard

Generated Model



Model Generation – Moving Load Definition

Activation of Slab Elements

1. To view the model in Front view

   Click on

2. Click on **"Select Single"** and

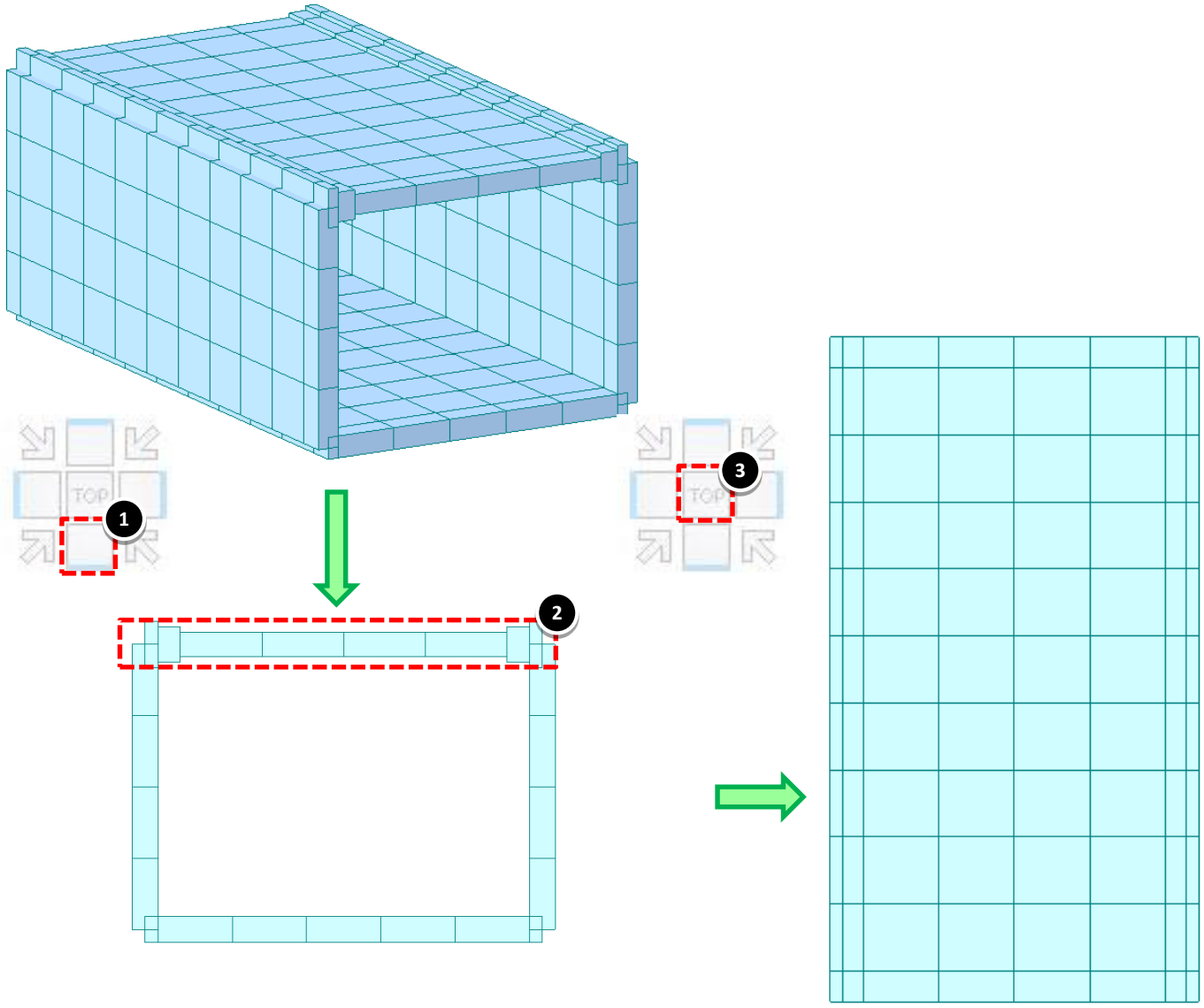
   Select the top portion as shown

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



Model Generation – Moving Load Definition

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

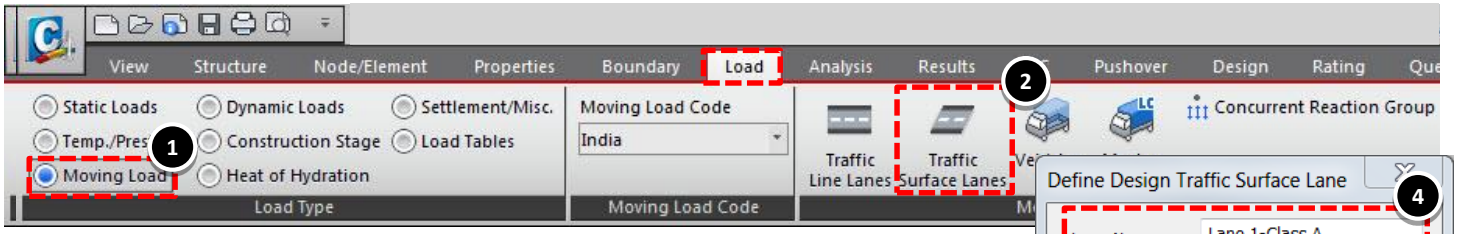
   Span Length > **5.35** m
6. Moving Direction > **Both**

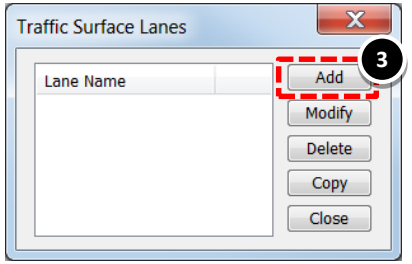
   Select Selection by "**2 Points**"

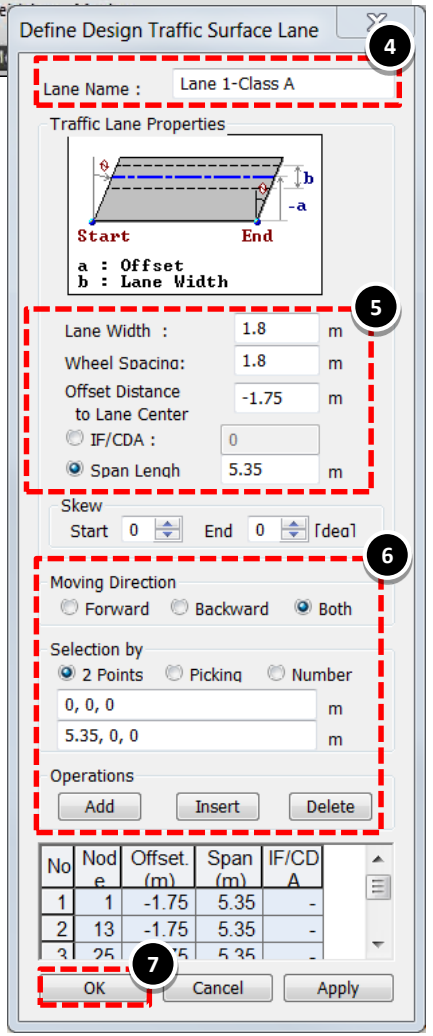
   Click on node no. **1**

   Click on Node no. **97**
7. Click "OK"

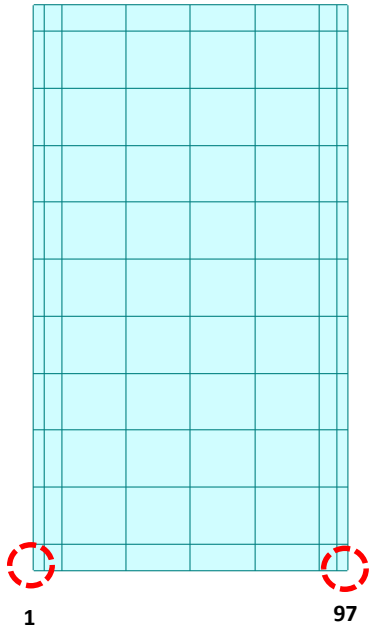
Similarly define remaining lanes as per the table







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



Step **01**

Model Generation – Moving Load Definition

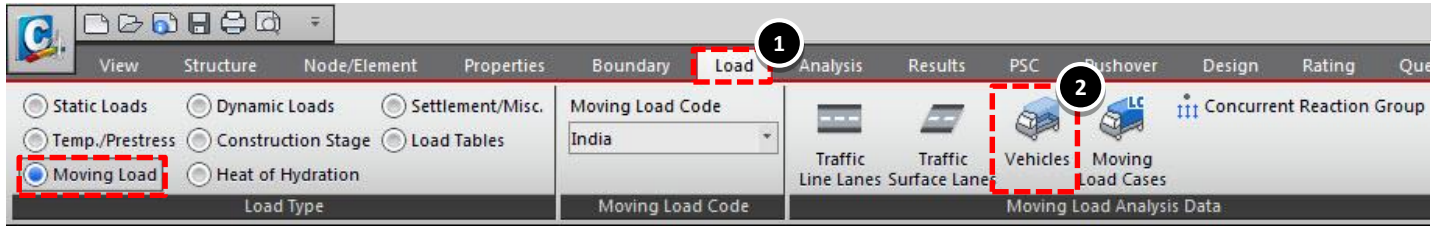
Define Vehicles

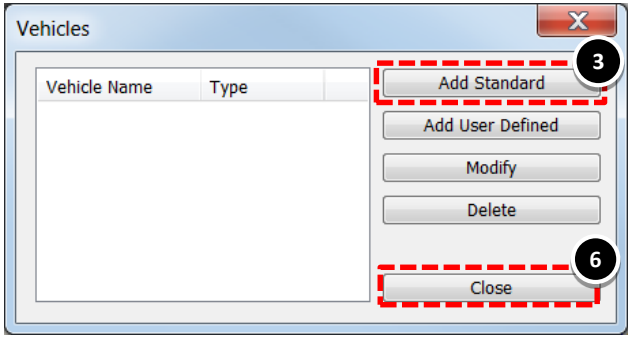
1. Go to “**Load**” > “**Moving Load**”
2. Click “**Vehicles**”
3. Click “**Add Standard**”
4. Select **“Standard Name”** > “**IRC:6-2000 Standard Load**”

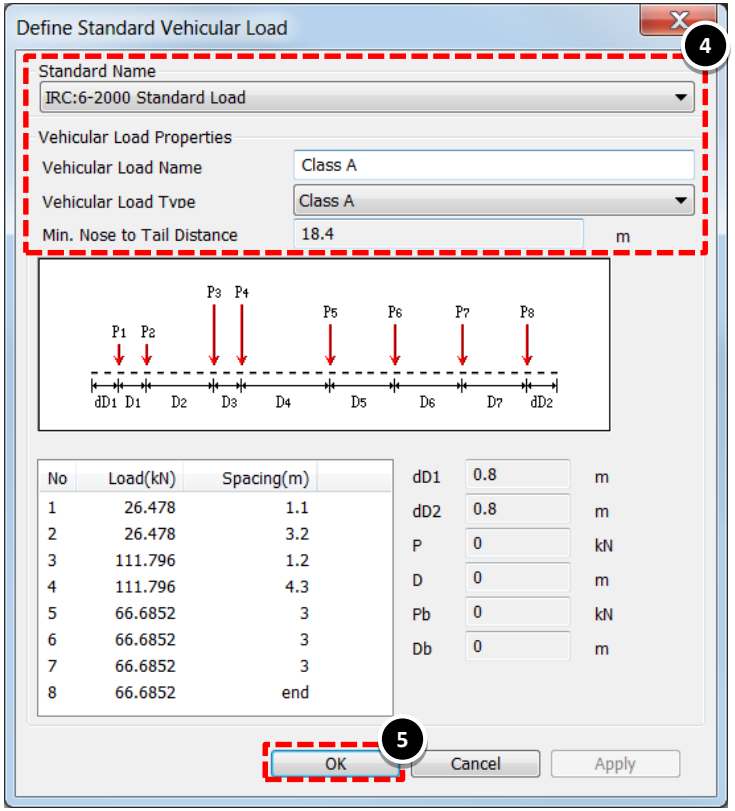
   Select Vehicular Load Type > “**Class A**”

   Click “**Apply**”

   Similarly define vehicle standards as **Class 70R**
5. Click “**OK**”
6. Click “**Close**”







Step **01**

Model Generation – Moving Load Definition

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**

   Uncheck "**Auto Live Load Combination**"
5. Click "**Add**" under Sub-Load Cases
6. In **Sub-Load Case**

   Enter "**Scale Factor" > 1**

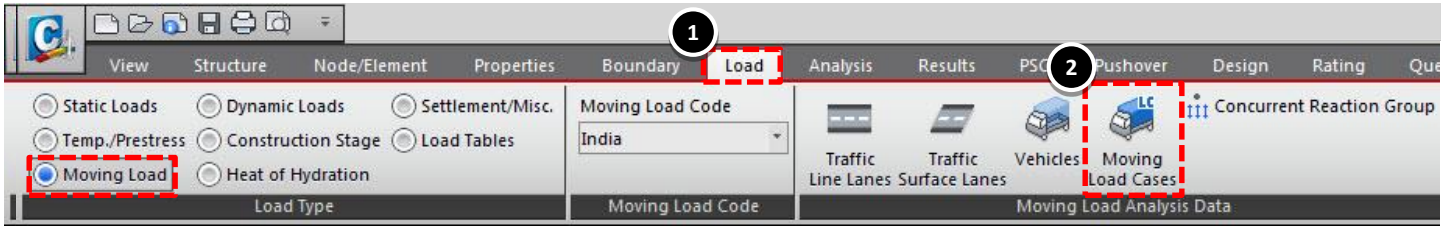
   Enter "**Minimum Loaded Lanes" > 1**

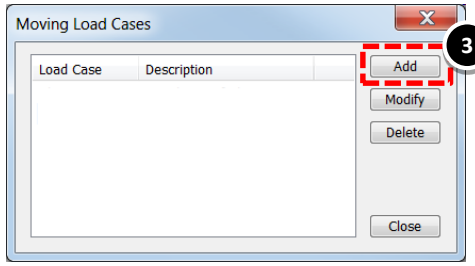
   Enter **"Maximum Loaded Lanes" >2**

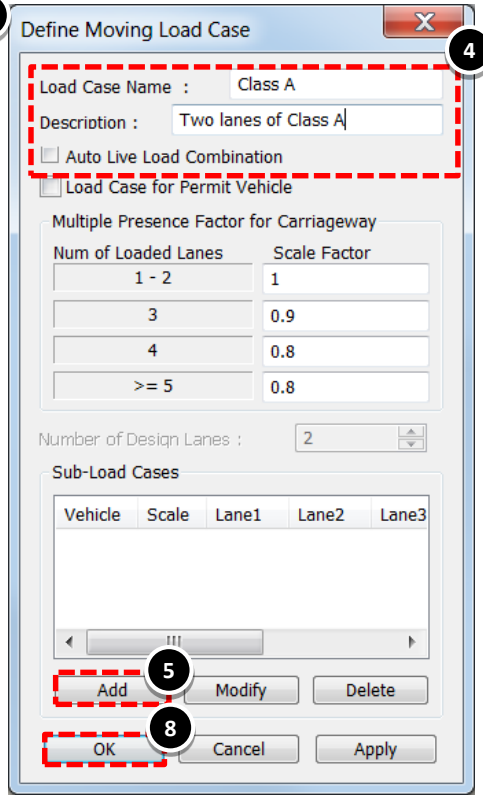
   Select **"Vehicle"** > **Class A**

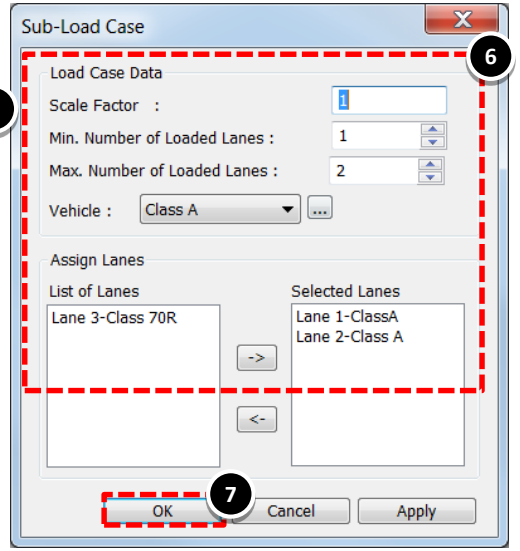
   Under List of Lanes Select **"Lane1- Class A"** and **"Lane2- Class A"**

   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.

Model Generation – Moving Load Definition

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**

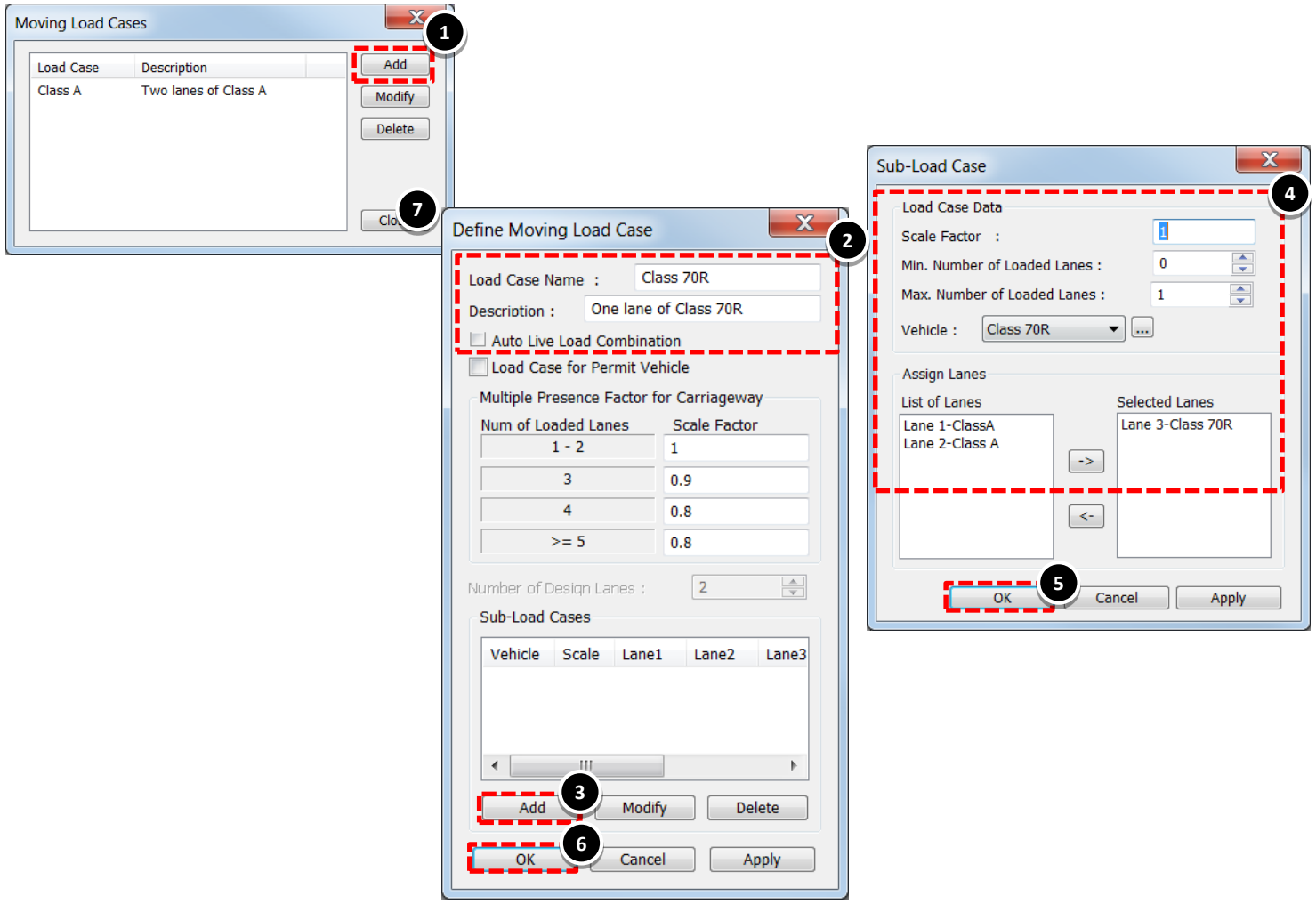
   Enter "**Minimum Loaded Lanes" > 1**

   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**"



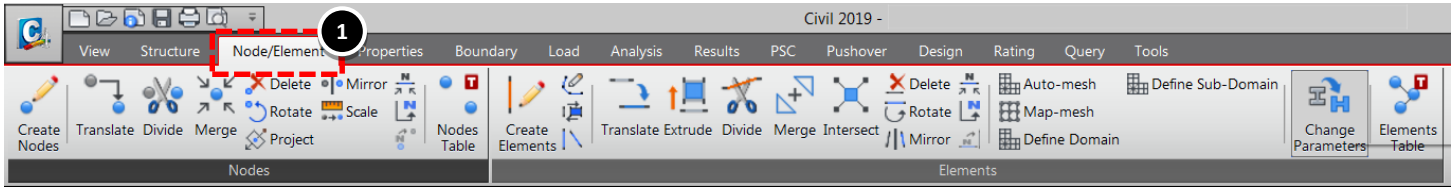
Note: To take into account of the wheel spacing and minimum clearance for different vehicles, the Auto Live Load Combination option can be unchecked and manually different moving load combinations can be created for the vehicles as per IRC 6:2000.

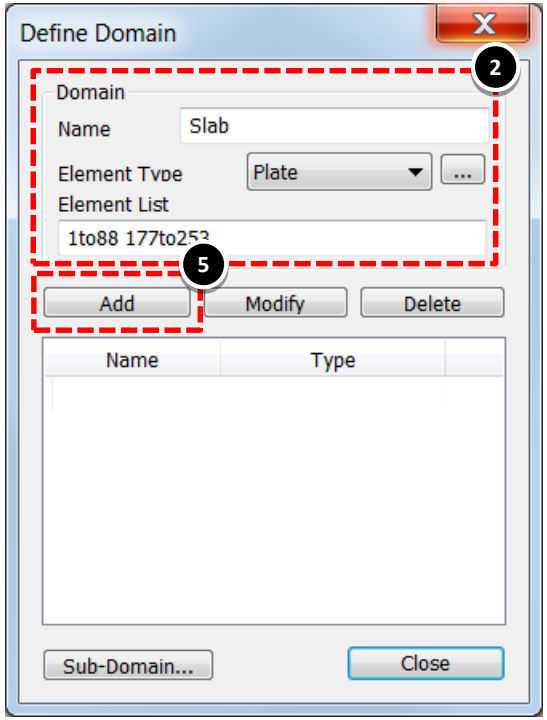
Step **01**

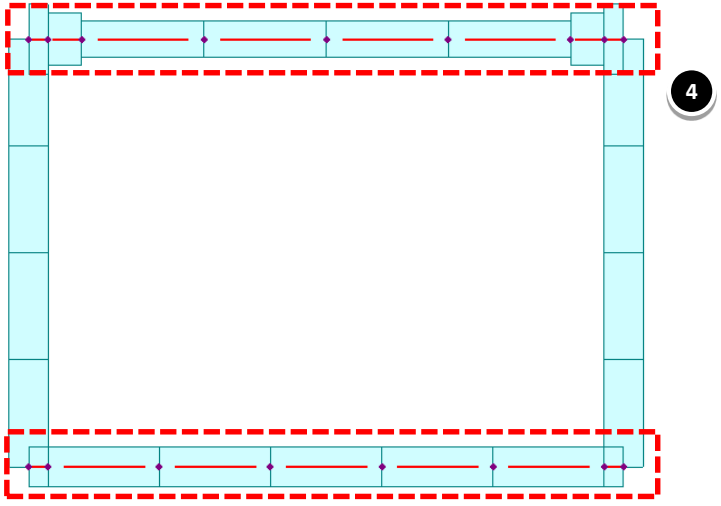
Model Generation – Domain Definition

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**





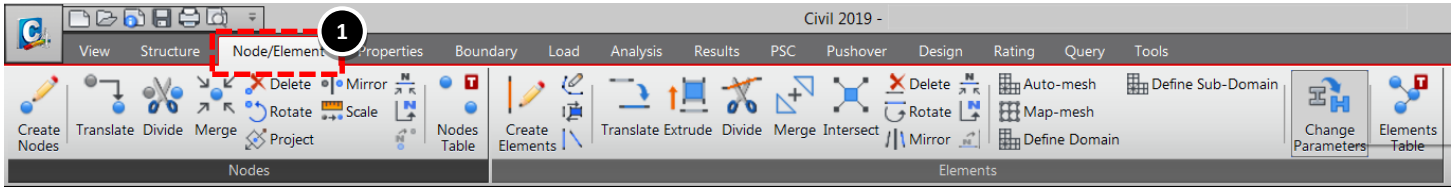


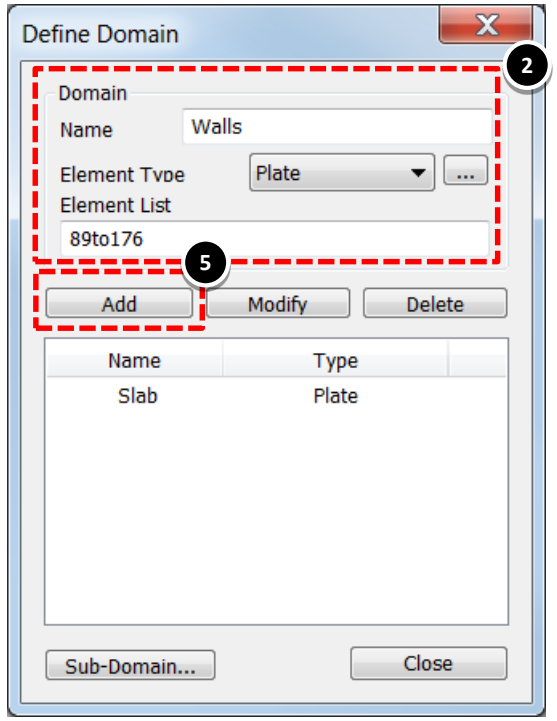
Model Generation – Domain Definition

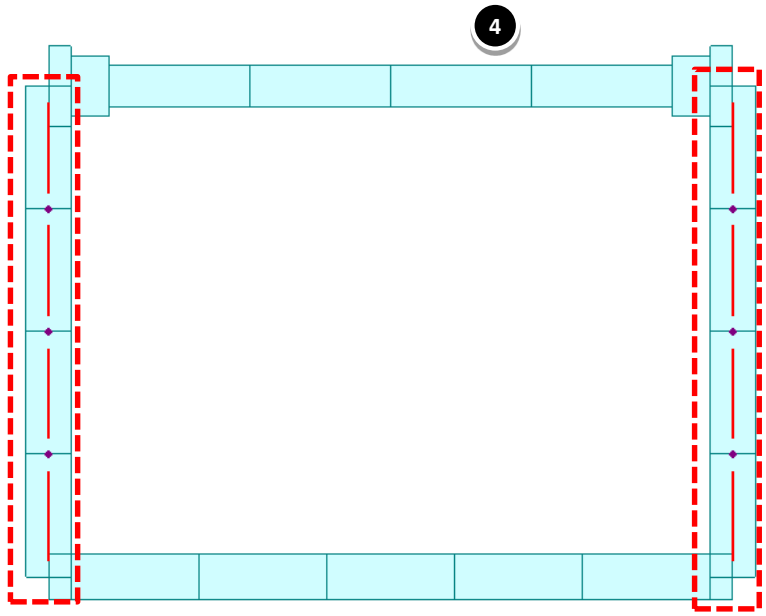
Domain for Walls

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



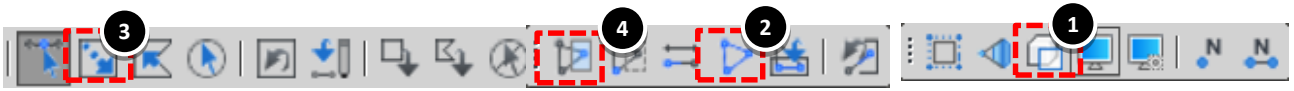




Model Generation – Domain Definition

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



Model Generation – Domain Definition

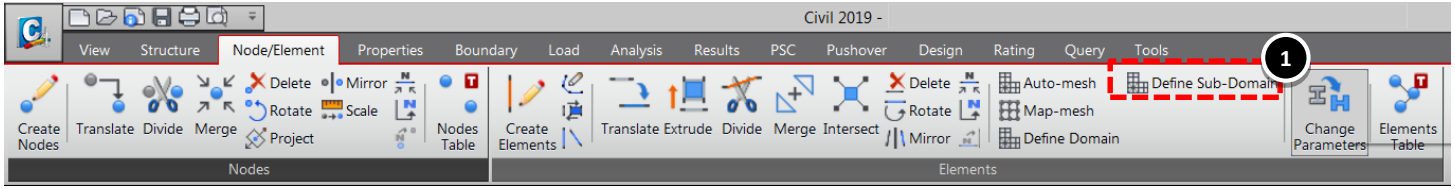
Sub-Domain for End slabs

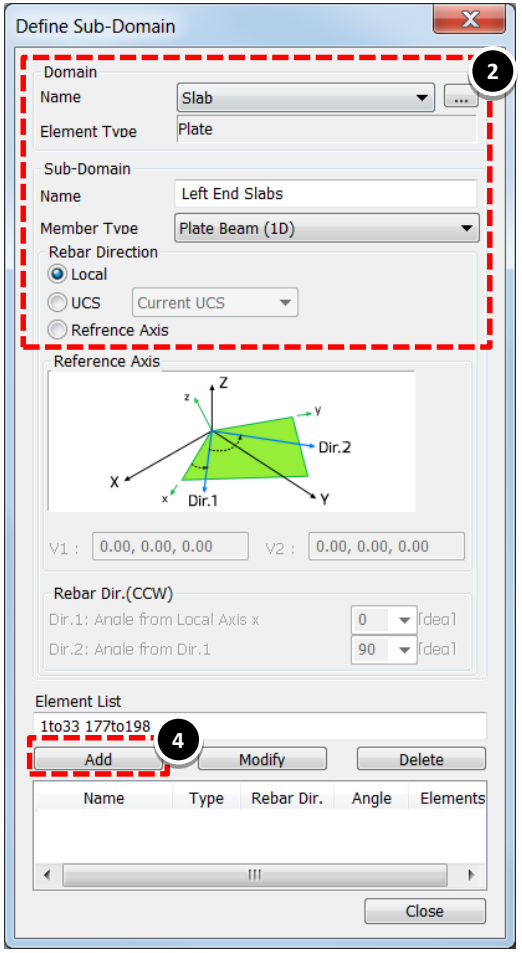
1. Go to Node/Elements > Define Sub-Domain
2. Domain Name > **Slabs**

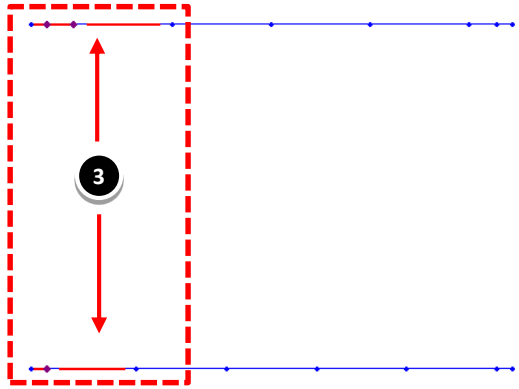
   Sub Domain Name > **Left End Slabs**

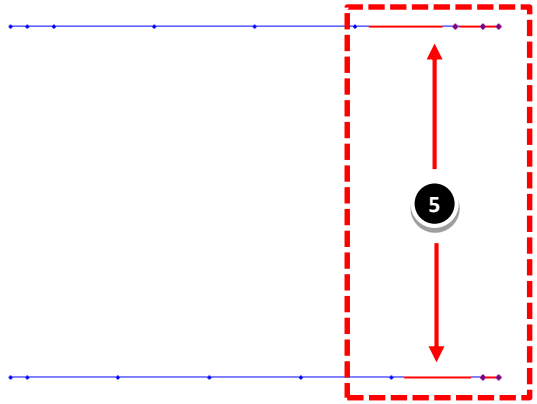
   Member Type > **Plate Beam (1D)**

   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**









Model Generation – Domain Definition

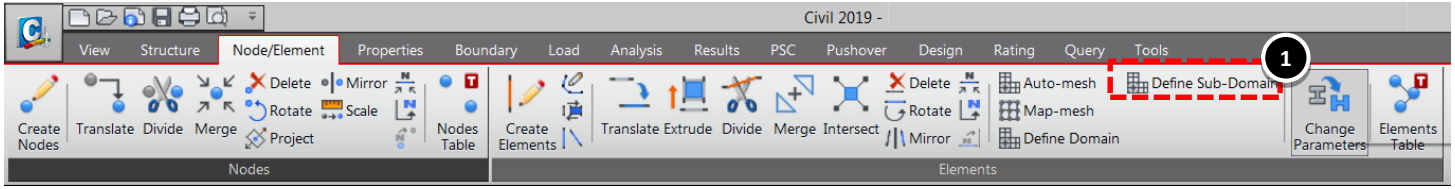
Sub-Domain for Mid slabs

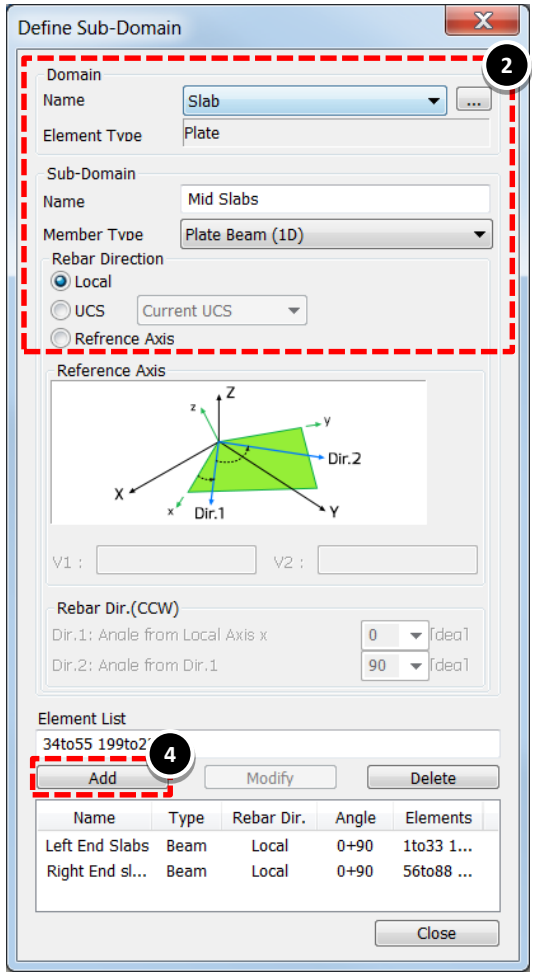
1. Go to Node/Elements > Define Sub-Domain
2. Domain Name > **Slabs**

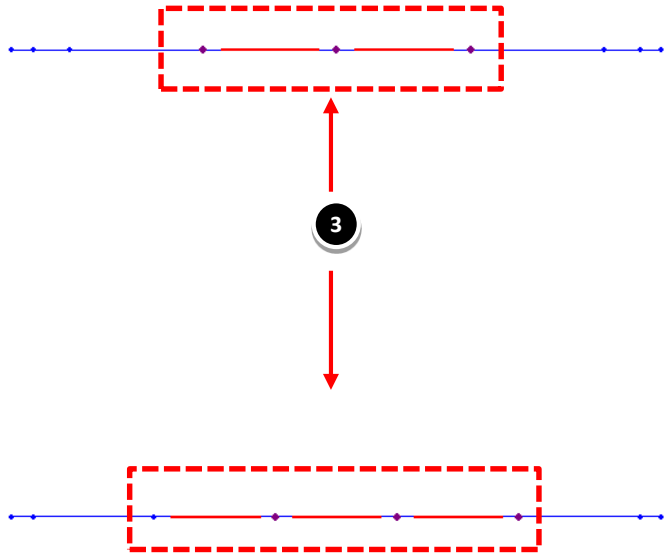
   Sub Domain Name > **Mid Slabs**

   Member Type > **Plate Beam (1D)**

   Rebar Direction > **Local**
3. Select top and bottom slab Elements as in image
4. Click on > **Add**



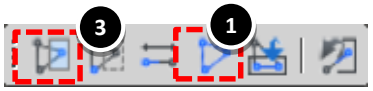


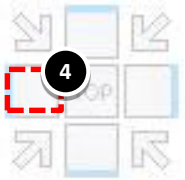


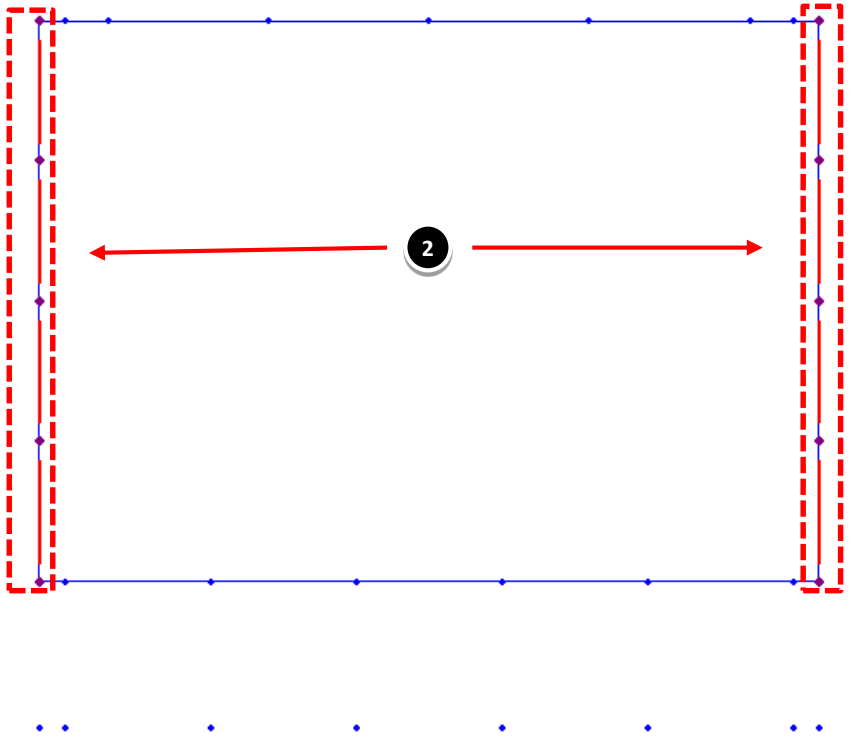
Model Generation – Domain Definition

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







Model Generation – Domain Definition

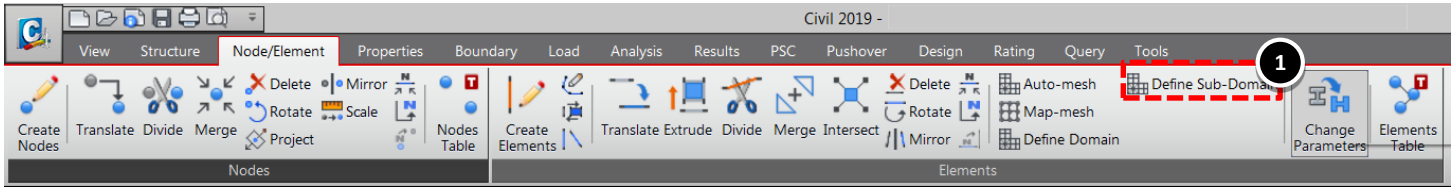
Sub-Domain for Top & Bottom walls

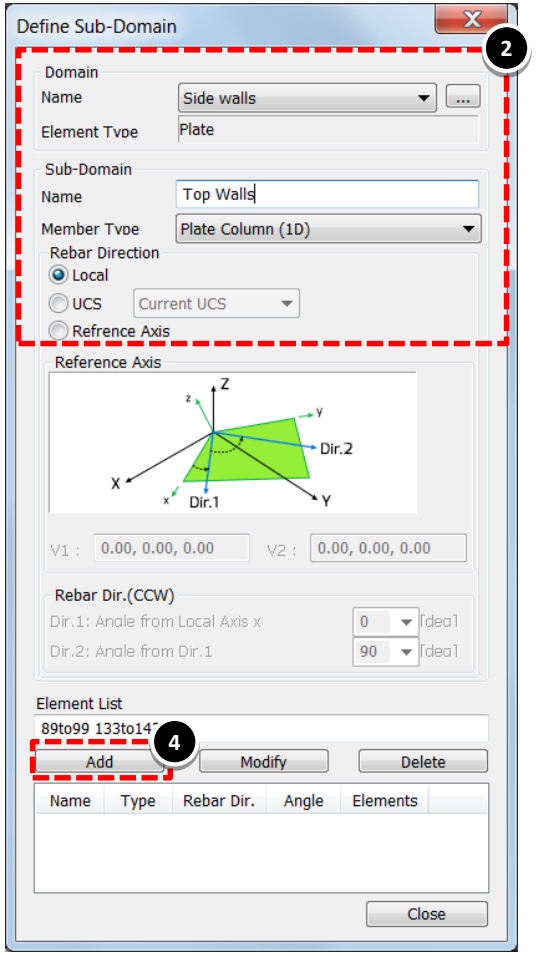
1. Go to Node/Elements > Define Sub-Domain
2. Domain Name > **Side walls**

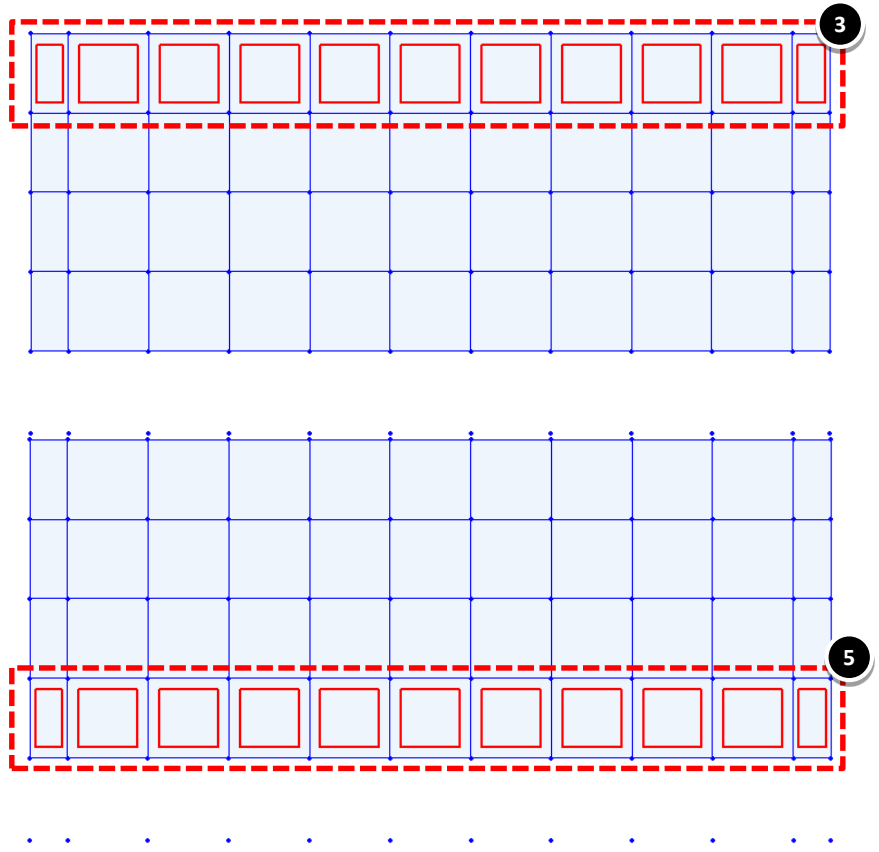
   Sub Domain Name > **Top walls**

   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**







Model Generation – Domain Definition

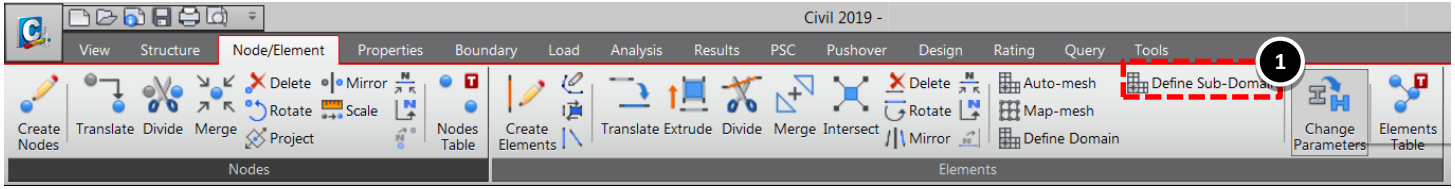
Sub-Domain for Mid walls

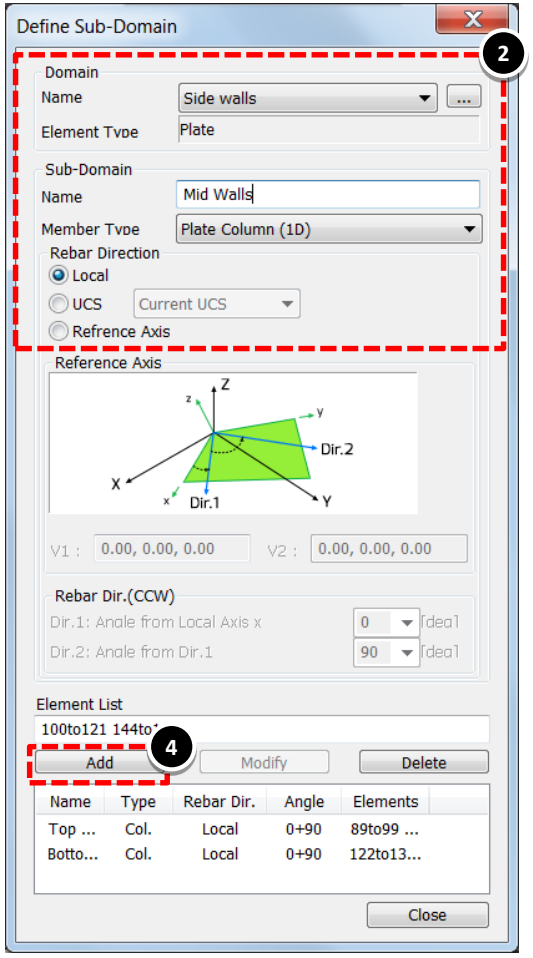
1. Go to Node/Elements > Define Sub-Domain
2. Domain Name > **Side walls**

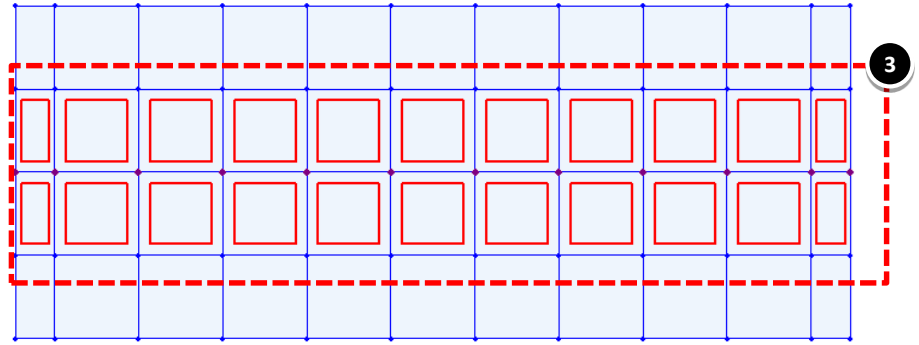
   Sub Domain Name > **Mid walls**

   Member Type > **Plate Column (1D)**

   Rebar Direction > **Local**
3. Select Mid wall Elements as in image
4. Click on > **Add**



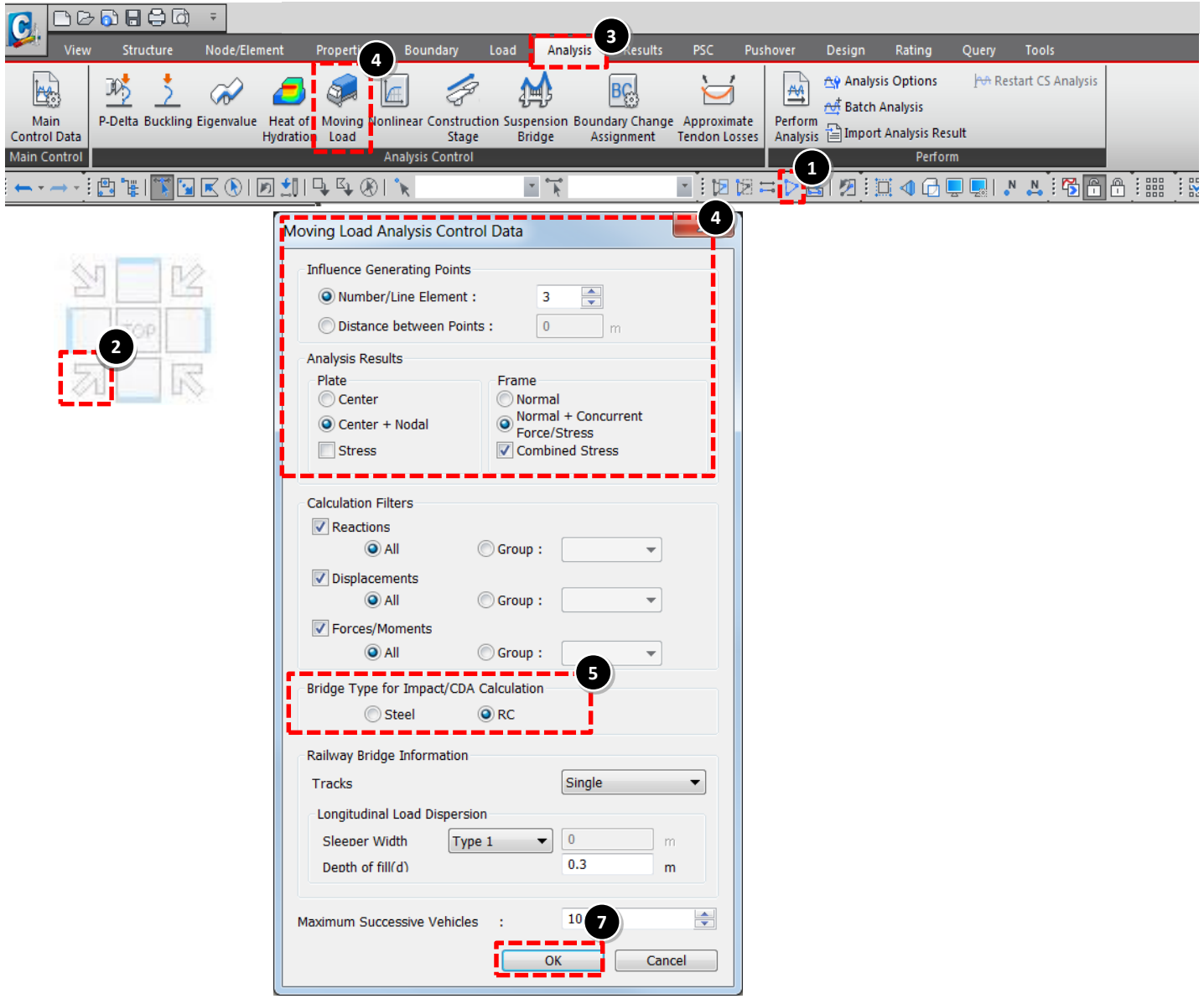




Analysis & Results – Analysis

Moving Load Analysis Control

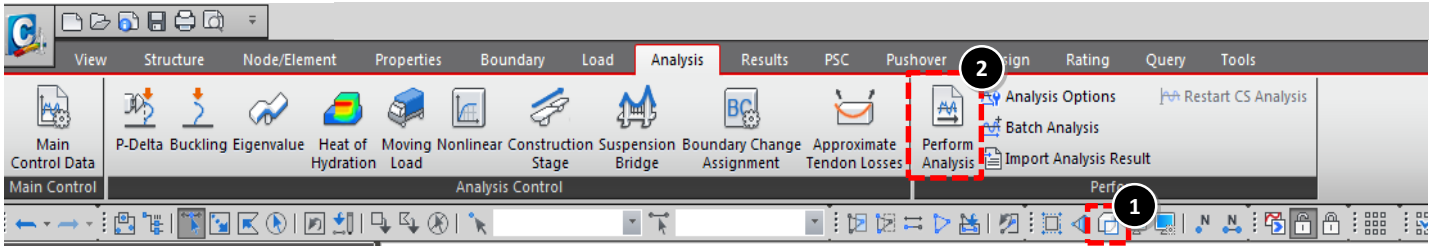
1. Click on > **"Activate All"**
2. Go to **Iso View**
3. Go to "**Analysis**" tab
4. Click "**Moving Load**"
5. In moving Load Analysis Control Data Window:
   - Enter Number/Line Elements > **3**
   - Under **Analysis Results > Frame**
   - Select "**Normal + Concurrent Force/Stress**"
   - Check "**Combined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

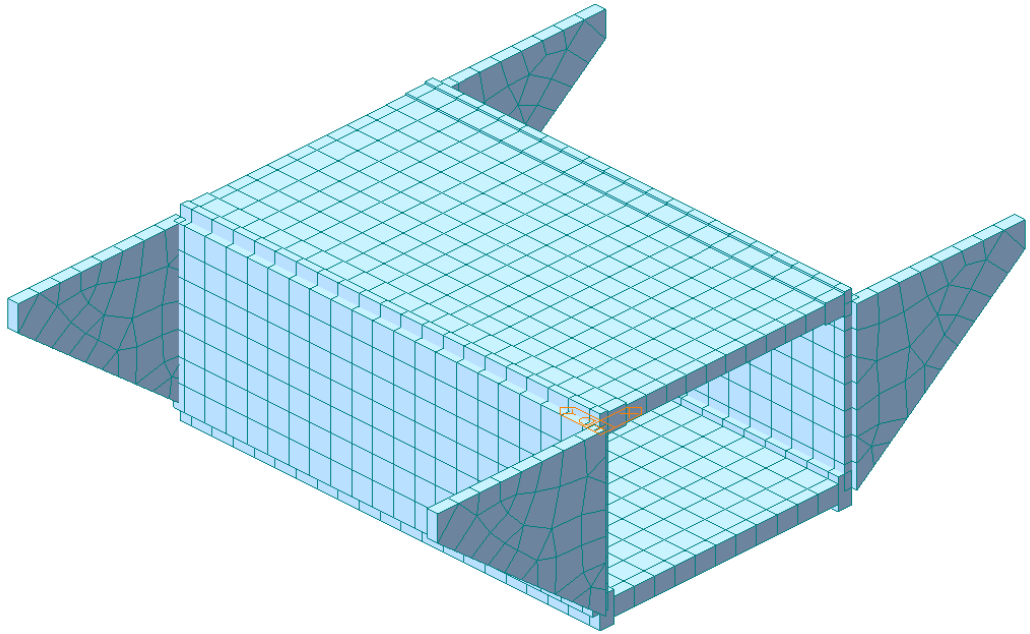


Analysis & Results – Analysis

Perform Analysis

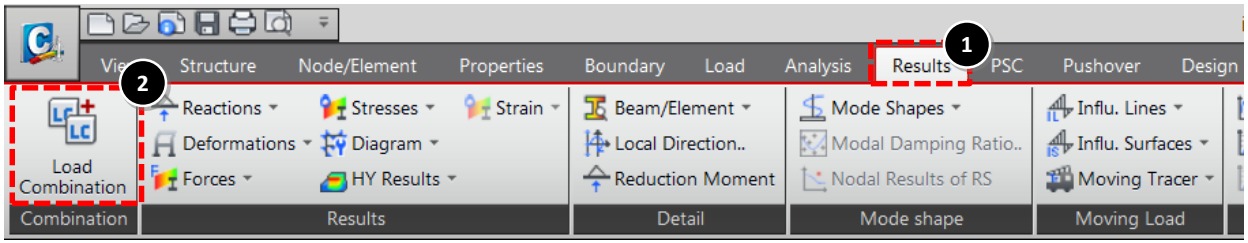
1. Click on > **Hidden**
2. Click "**Perform Analysis**" to Run Analysis

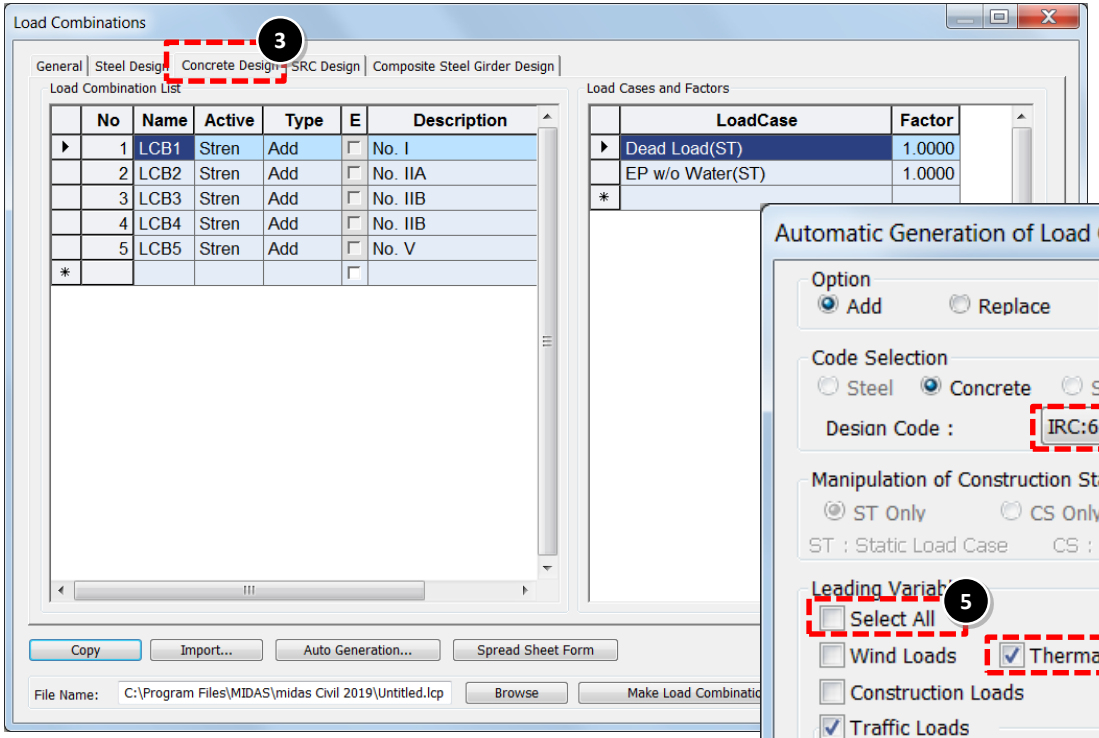


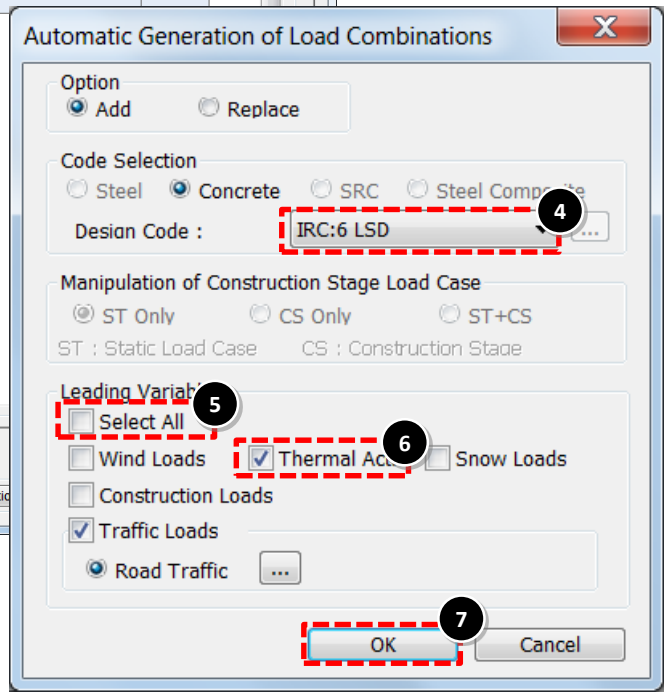


Auto Generation of LCB

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.**







Analysis & Results – Results

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”**

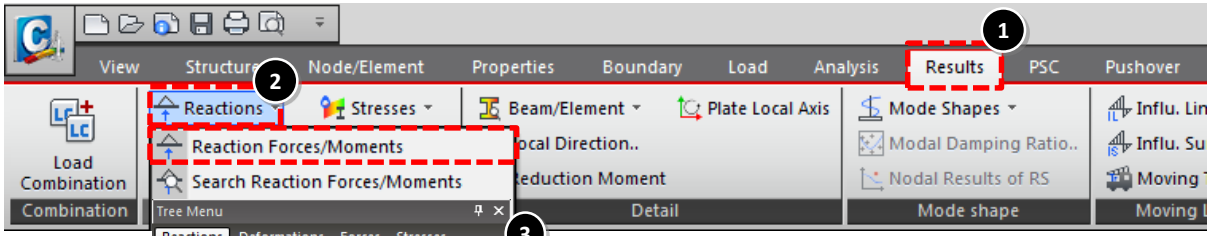
   Click “**Apply**”
4. To View results in Tabular format,

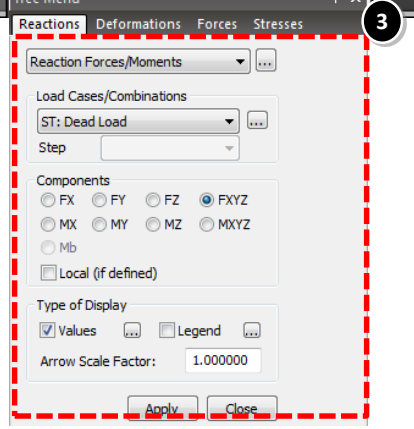
   Go to **“Result”** > **“Results Tables”>**
5. “**Reactions**”

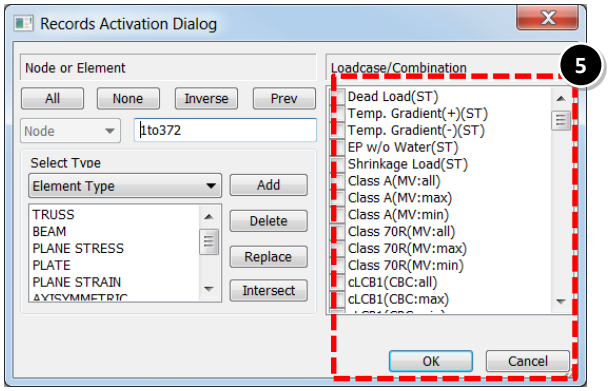
   Select “**Load Cases/Combination**”
6. Click “**OK**”

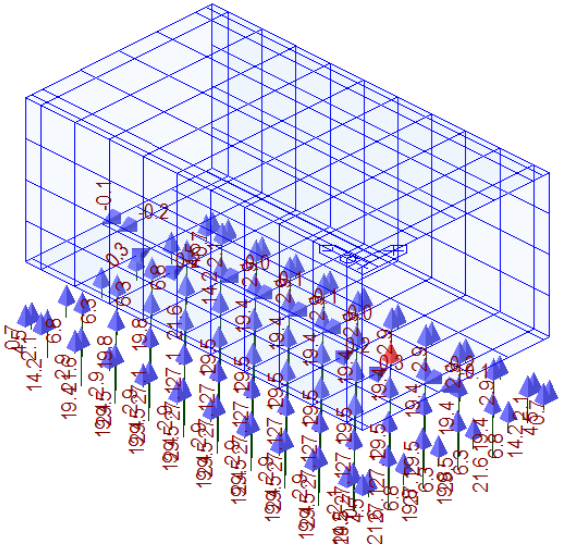
   Check the values in new window

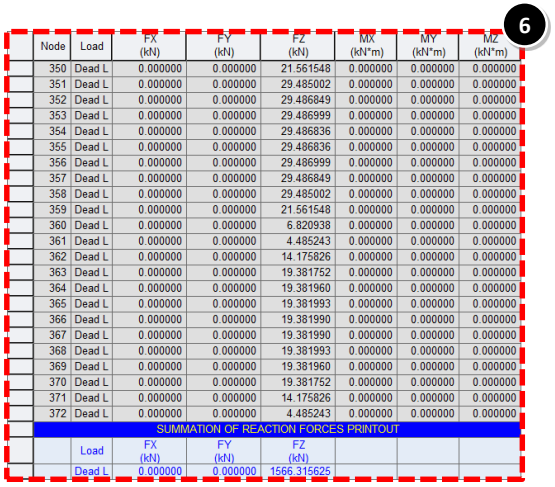
   “**Result-[Reaction]**”







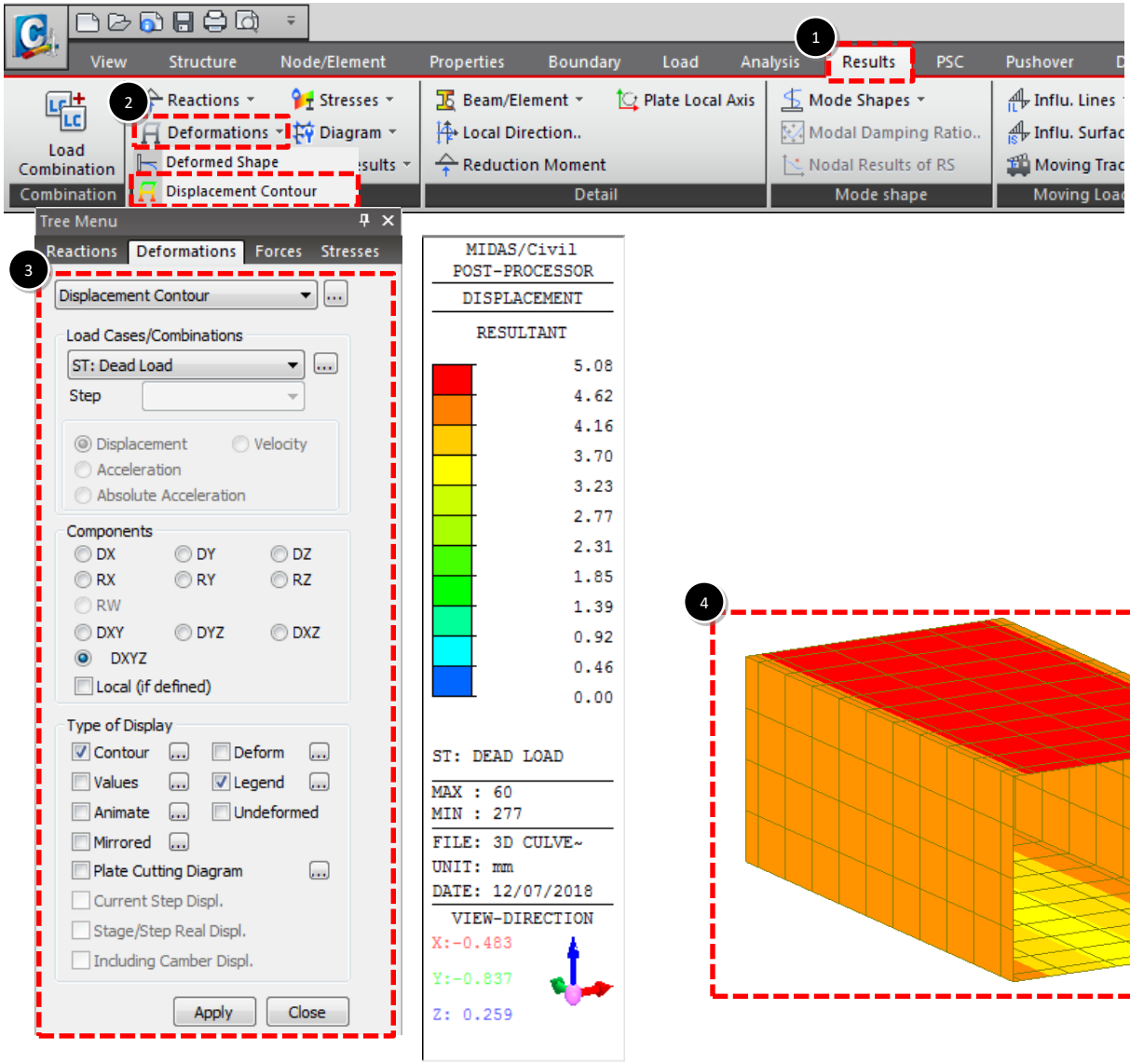




Analysis & Results – Results

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



Note: By Invoking" [...] " the tables of any component of result and load cases can be checked.

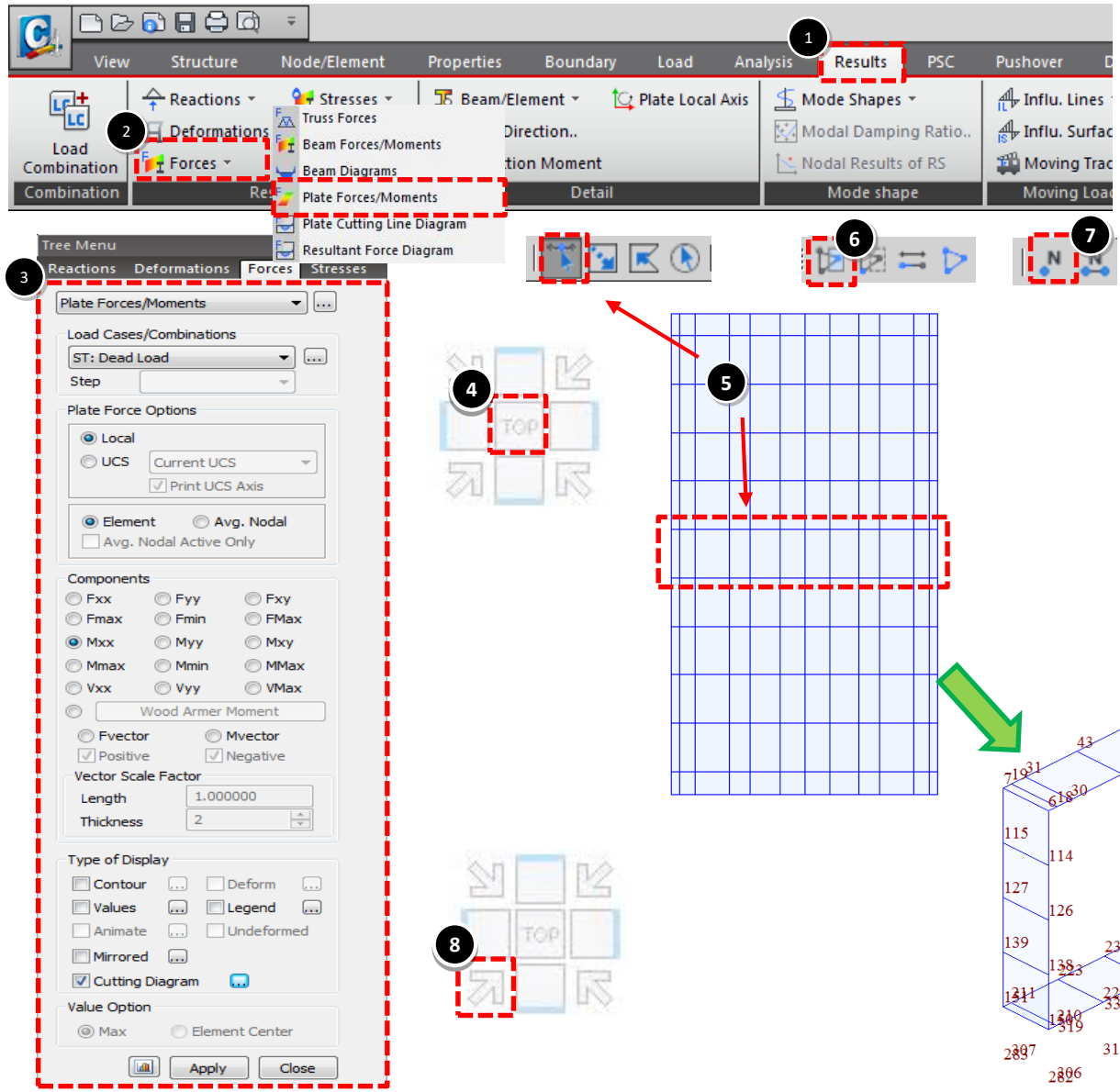
Analysis & Results – Results

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**"



Analysis & Results – Results

Plate Cutting Diagram

1. 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**

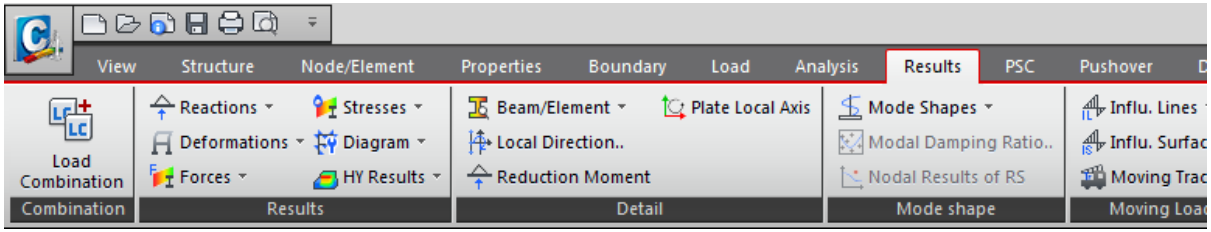
   Enter Name: **Right**
   Click in **Pnt1:** text box
   Click node **102** then **198** in model
   Click on **Add**

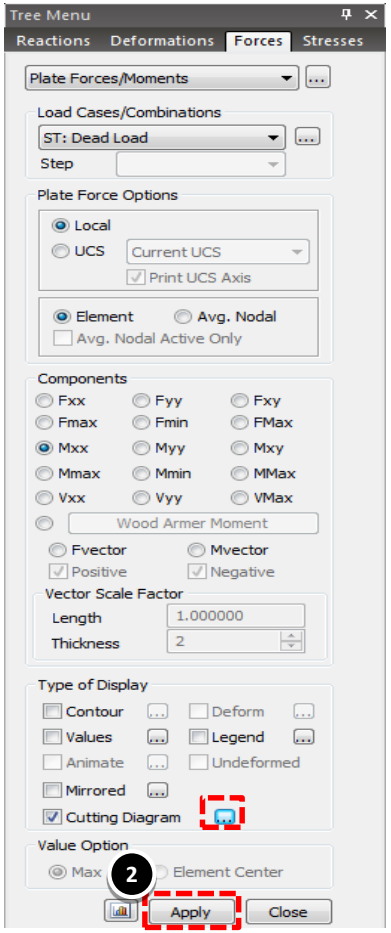
   Enter Name: **Bottom**
   Click in **Pnt1:** text box
   Click node **150** then **198** model
   Click on **Add**

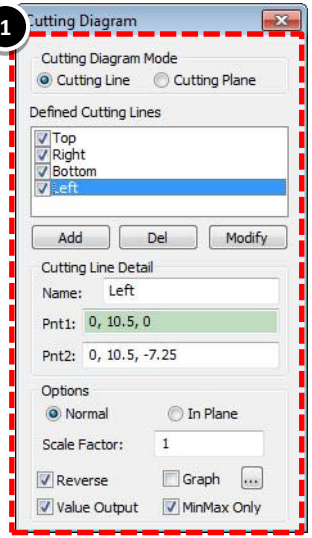
   Enter Name: **Left**
   Click in **Pnt1:** text box
   Click node **150** then **6** in model
   Click on **Add**

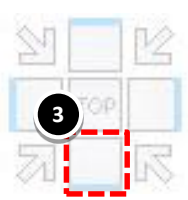
2. Click > **"Reverse"**
3. Click > **"Apply"**
4. Click on **"Front View"**

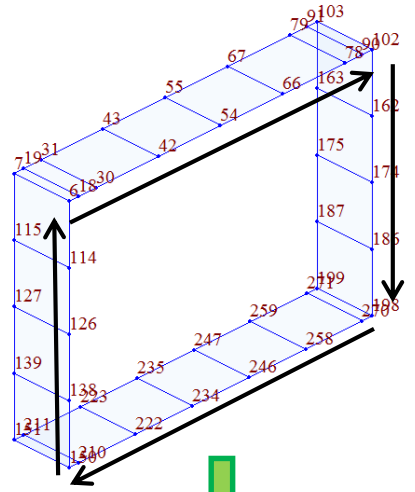
   See the cutting diagram in the "**Model View**" window

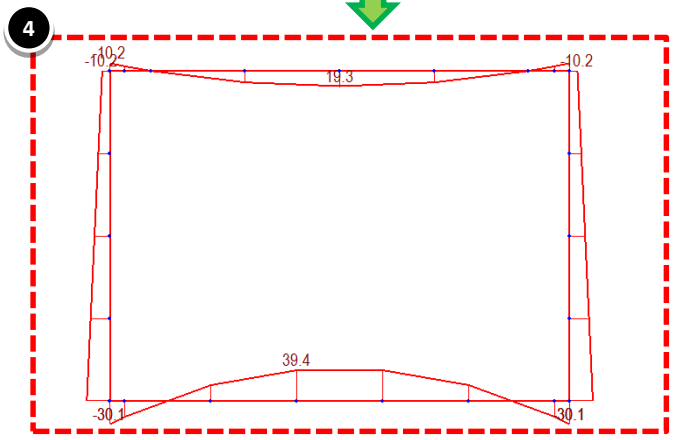








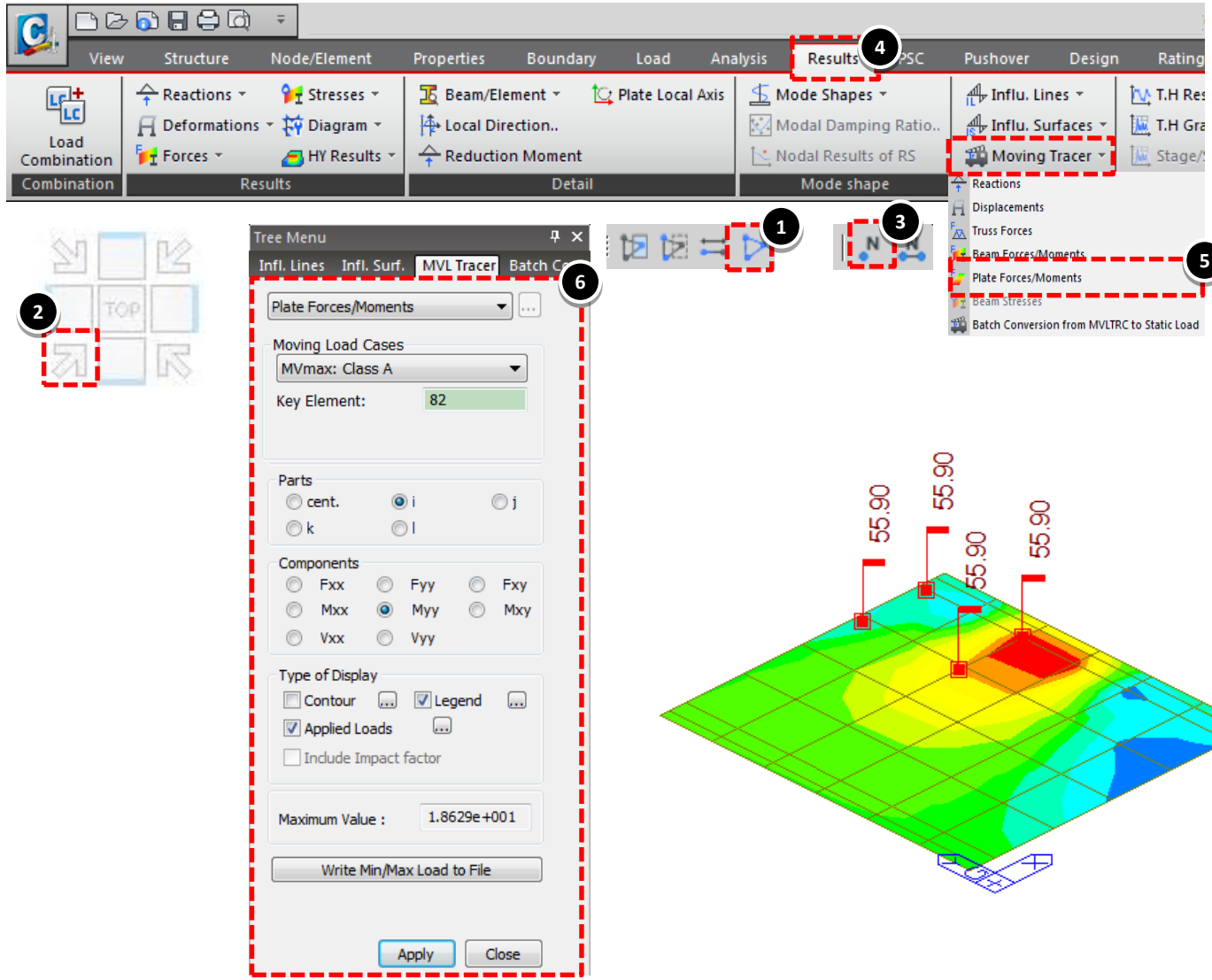




Analysis & Results – Results

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



Analysis & Results – Results

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)**

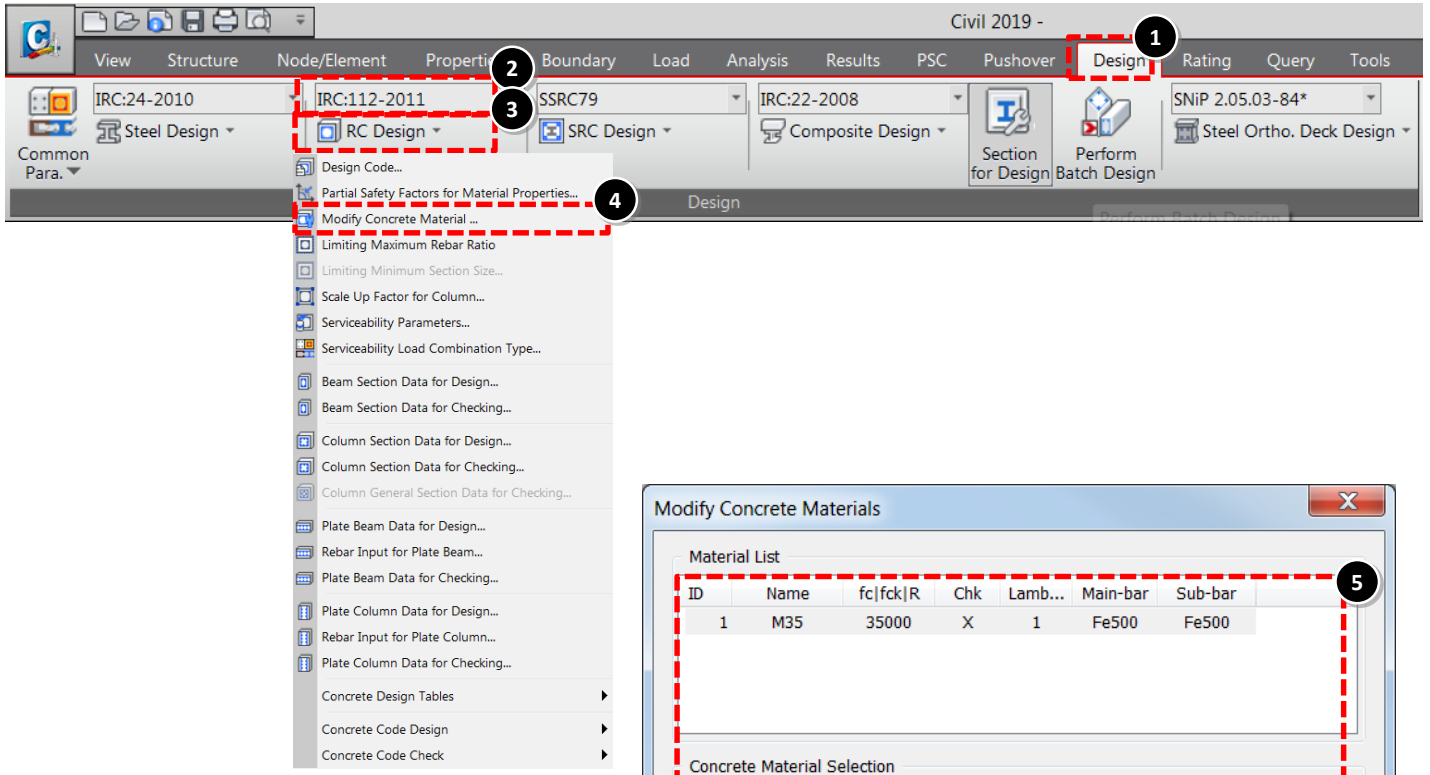
   Select Grade: **M35**

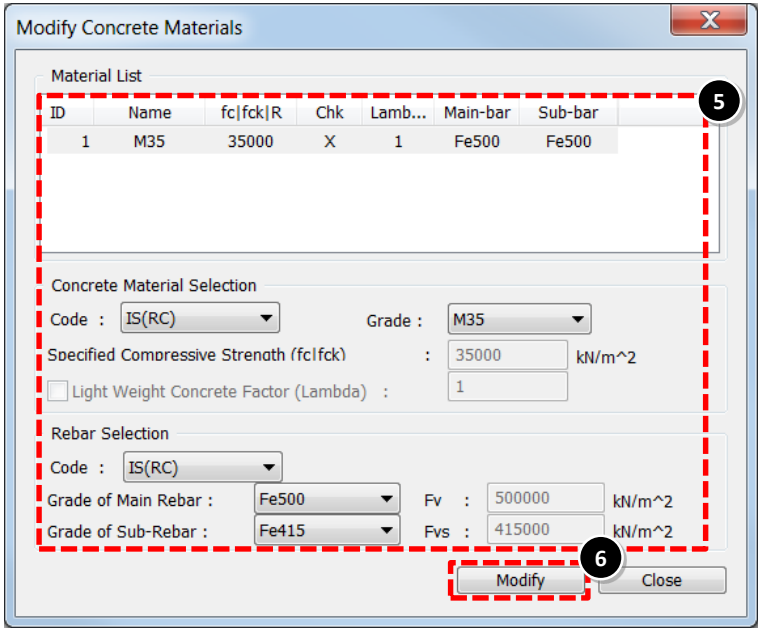
   Under Rebar Selection

   Select Code: **IS(RC)**

   Select Grade of Main Rebar: **Fe500**

   Select Grade of Sub-Rebar: **Fe415**
6. Click **Modify** & **Close**





Design – Plate Beam Data for Design

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**

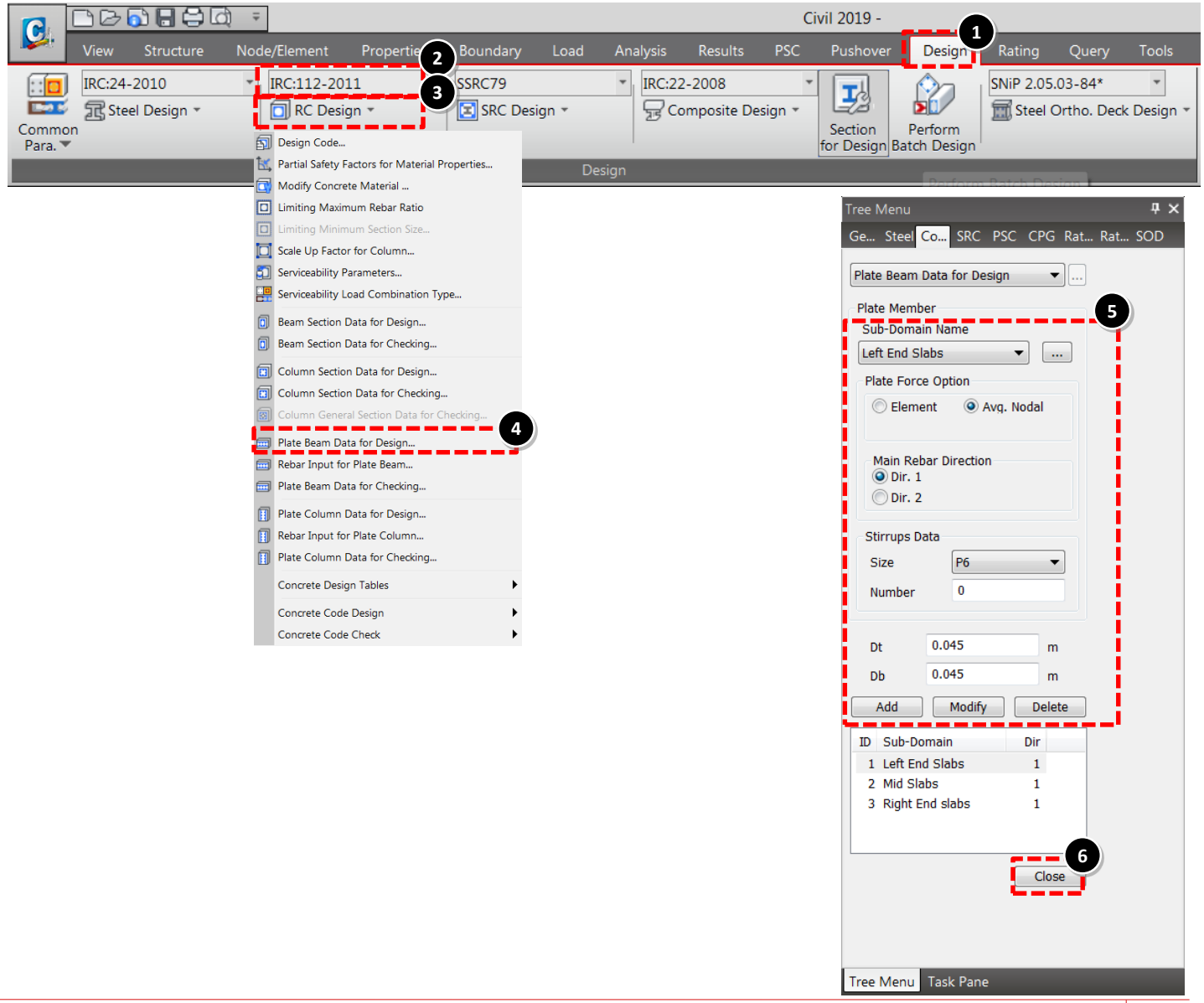
   Plate Force Option > **Avg. Nodal**

   Main Rebar Direction > **Dir. 1**

   Dt > **0.045 m**

   Dt > **0.045 m**

   Click on > **Add**
6. Click on > **Close**
7. Repeat **Step 5** to add Right End and Mid Slabs



Design – Plate Column Data for Design

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**

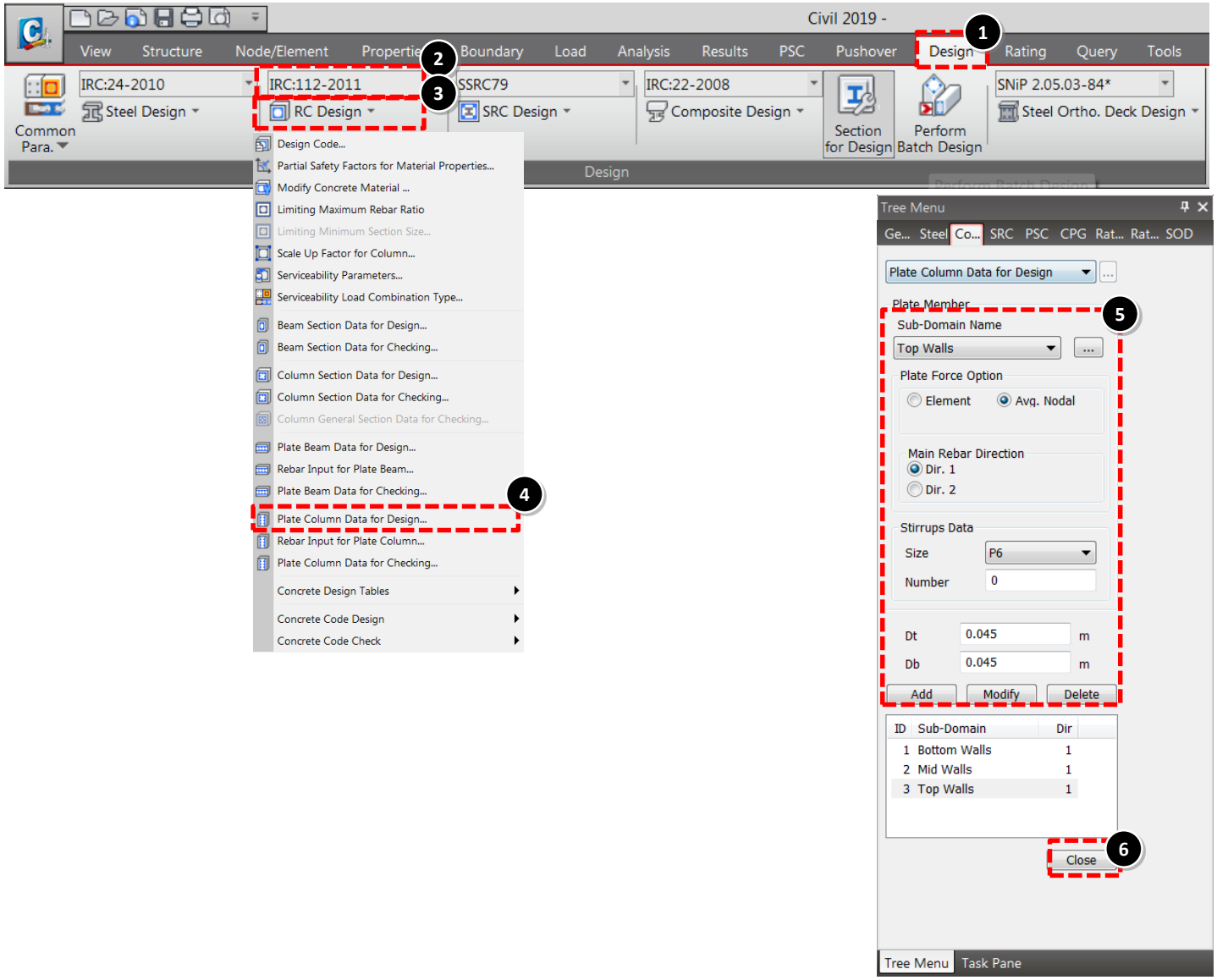
   Plate Force Option > **Avg. Nodal**

   Main Rebar Direction > **Dir. 1**

   Dt > **0.045 m**

   Dt > **0.045 m**

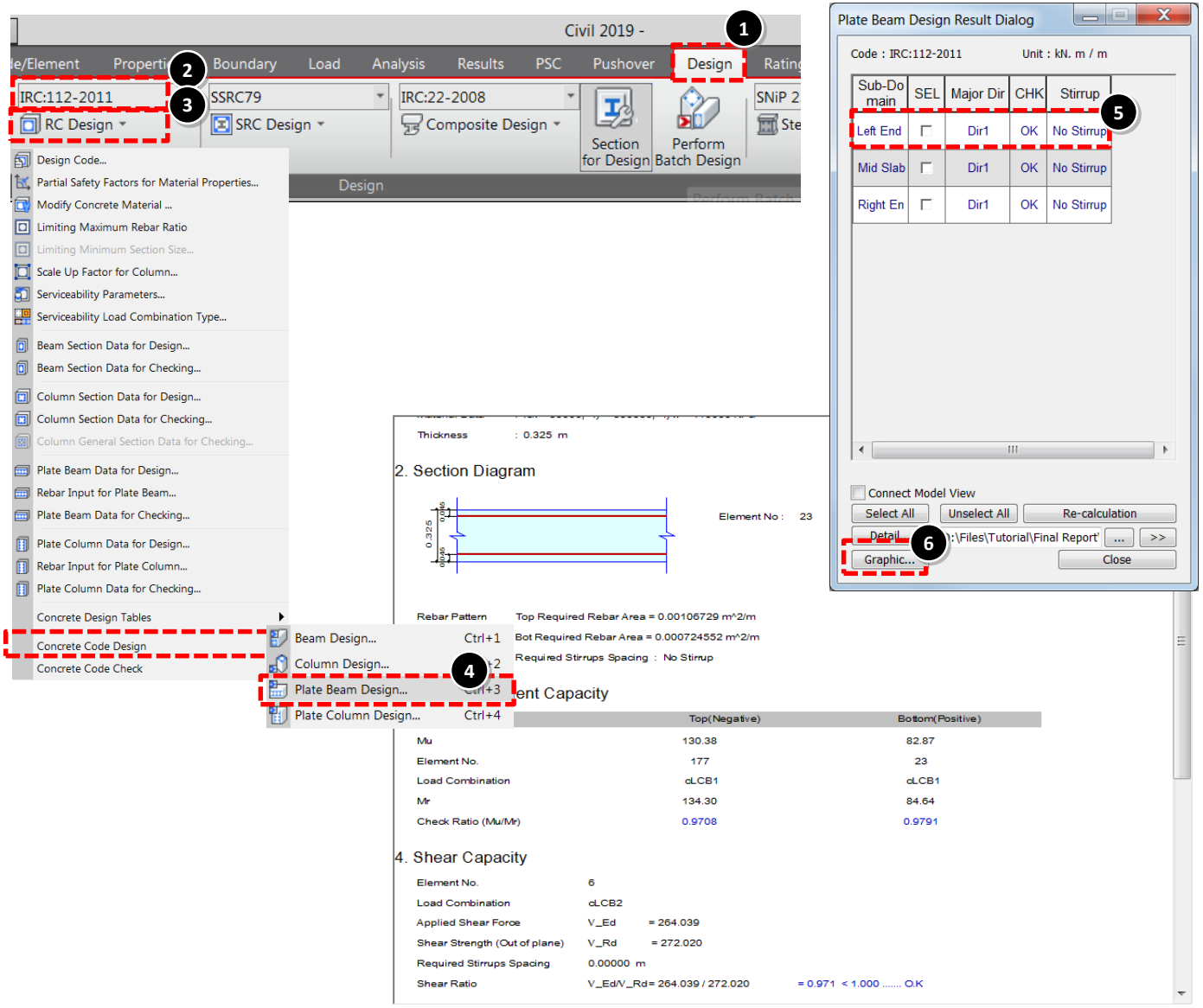
   Click on > **Add**
6. Click on > **Close**
7. Repeat **Step 5** to add Right End and Mid Slabs



Design – Plate Beam Design

Plate Beam Design

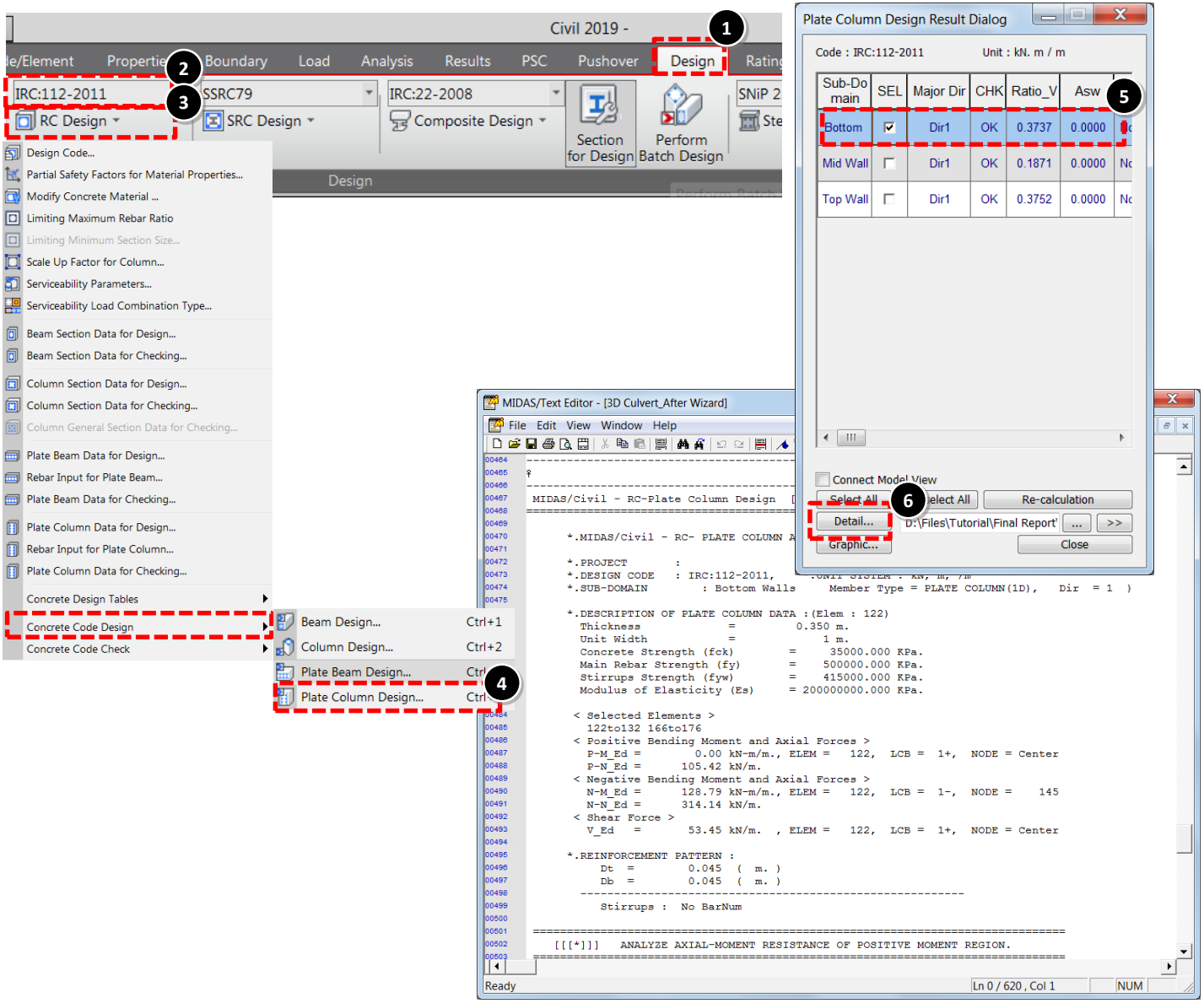
1. Go to "**Design**" tab
2. Select "**IRC 112-2011**"
3. Click "**RC Design**"
4. Select Concrete Code Design > **Plate Beam Design**
5. Select The Left End sub-Domain
6. Click on > **Graphic**



Design – Plate Column Design

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 – Rebar Input for Beam

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**

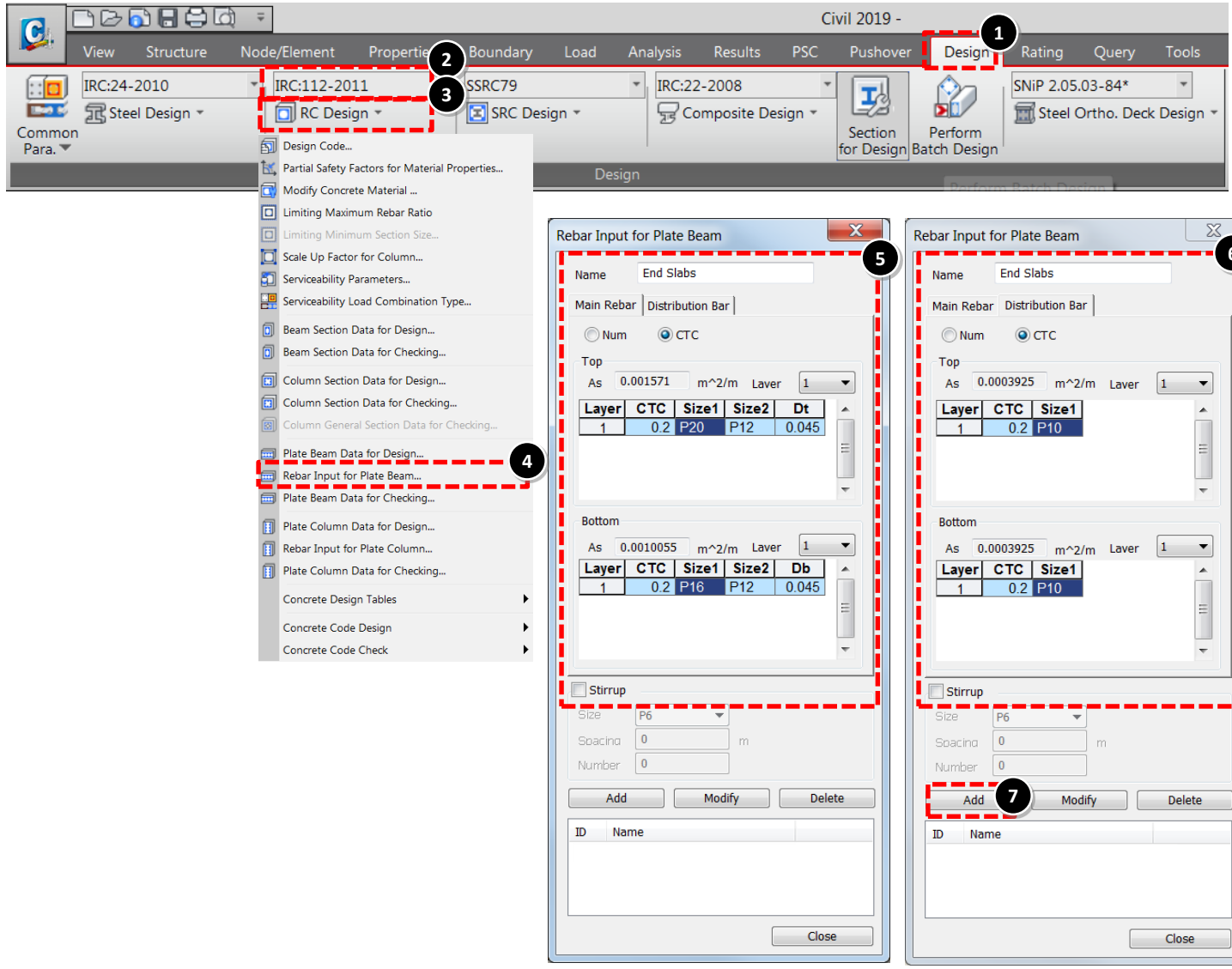
6. Under Distribution Rebar > **CTC** > **Top**

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

   Bottom >

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

7. Click on > **Add**



Design – Rebar Input for Beam

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**

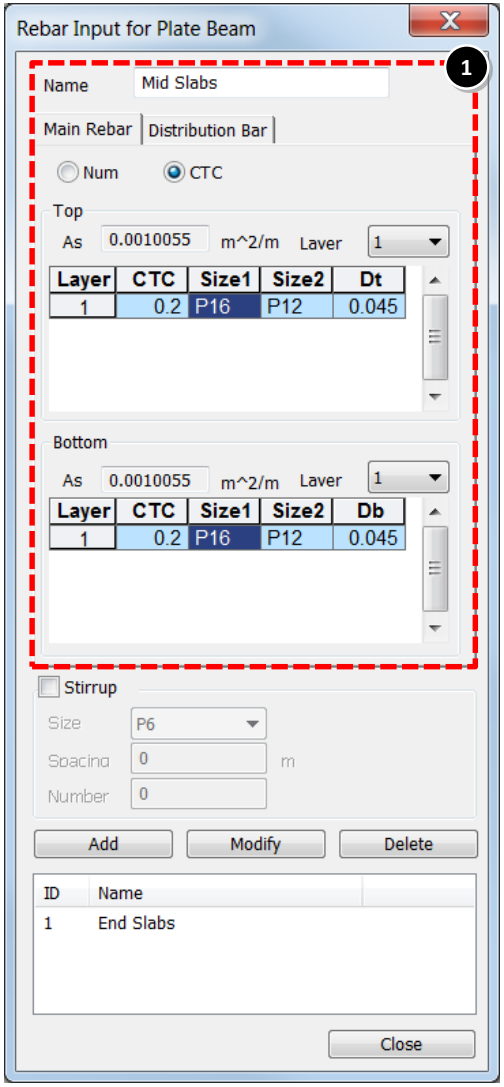
2. Under Distribution Rebar > **CTC** > **Top**

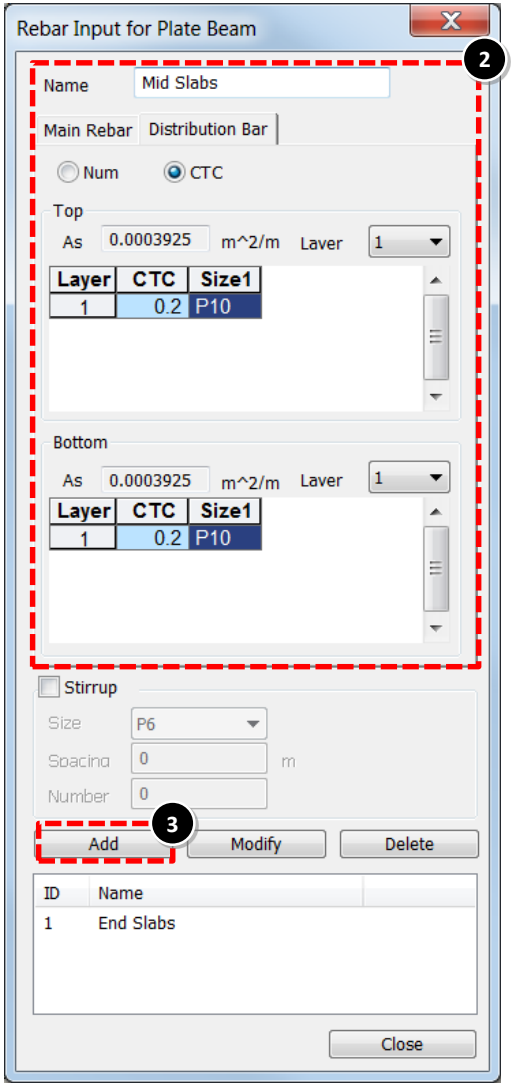
   CTC > 0.2; Size 1 > **P10**

   Bottom >

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

3. Click on > **Add**





Design – Rebar Input for Column

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

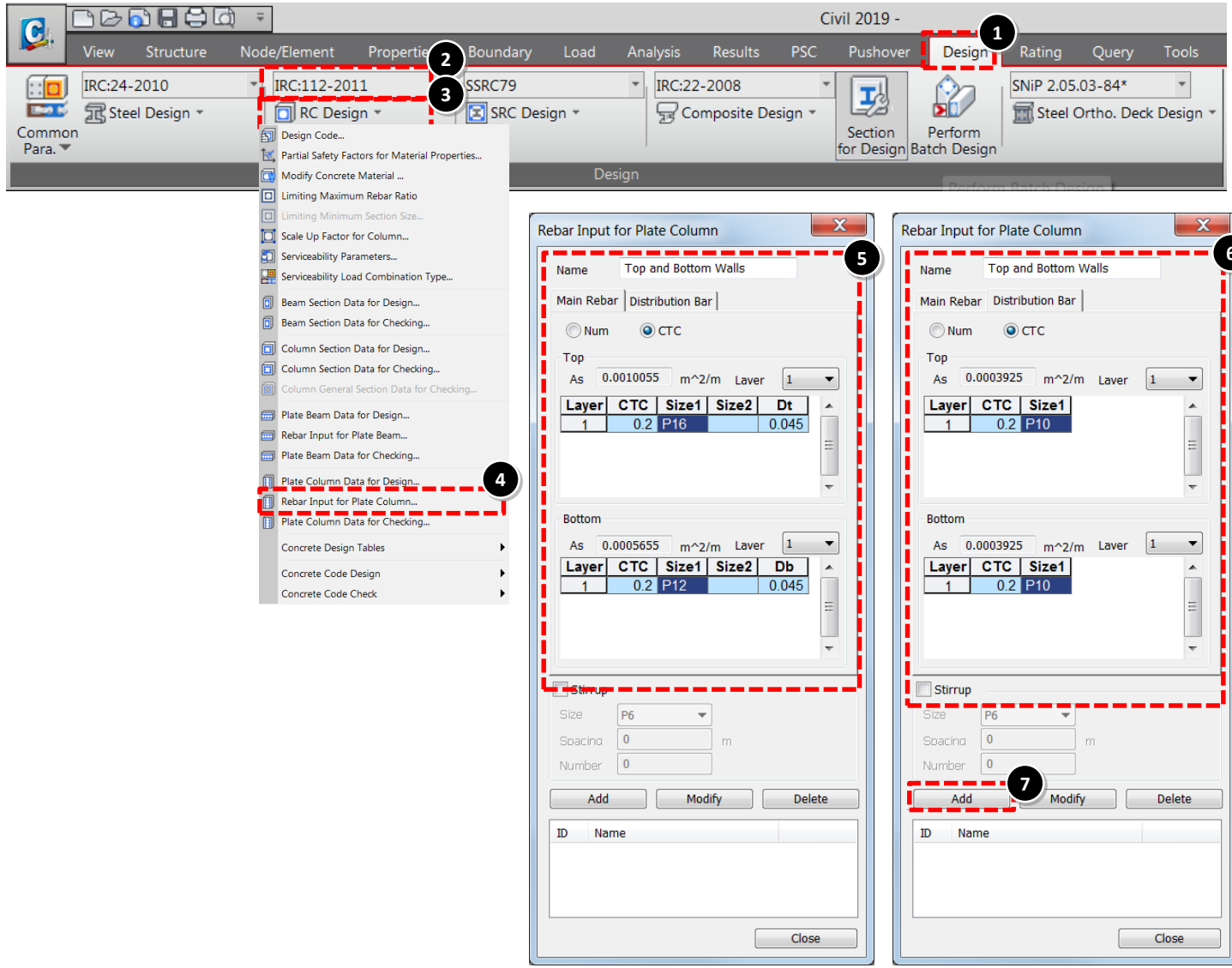
   Dt > 0.045
6. Under Distribution Rebar > **CTC** >

   Top

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

   Bottom >

   CTC > 0.2; Size 1 > **P10**
7. Click on > **Add**



Design – Rebar Input for Column

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**

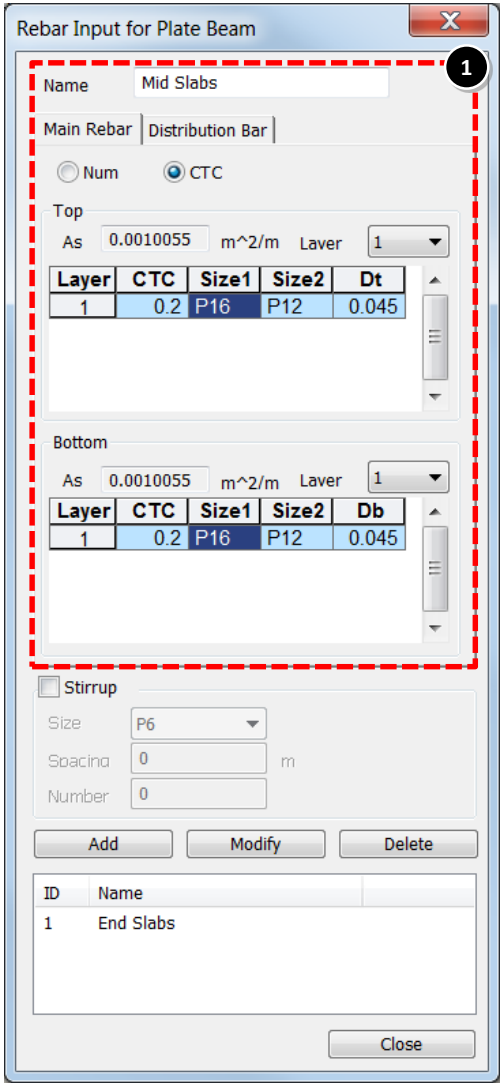
2. Under Distribution Rebar > **CTC** > **Top**

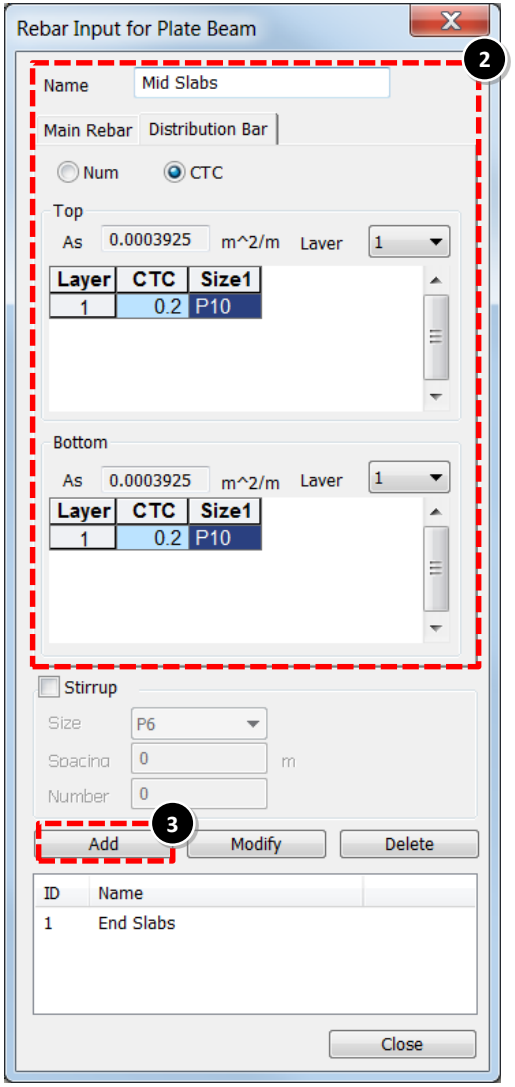
   CTC > 0.2; Size 1 > **P10**

   Bottom >

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

3. Click on > **Add**

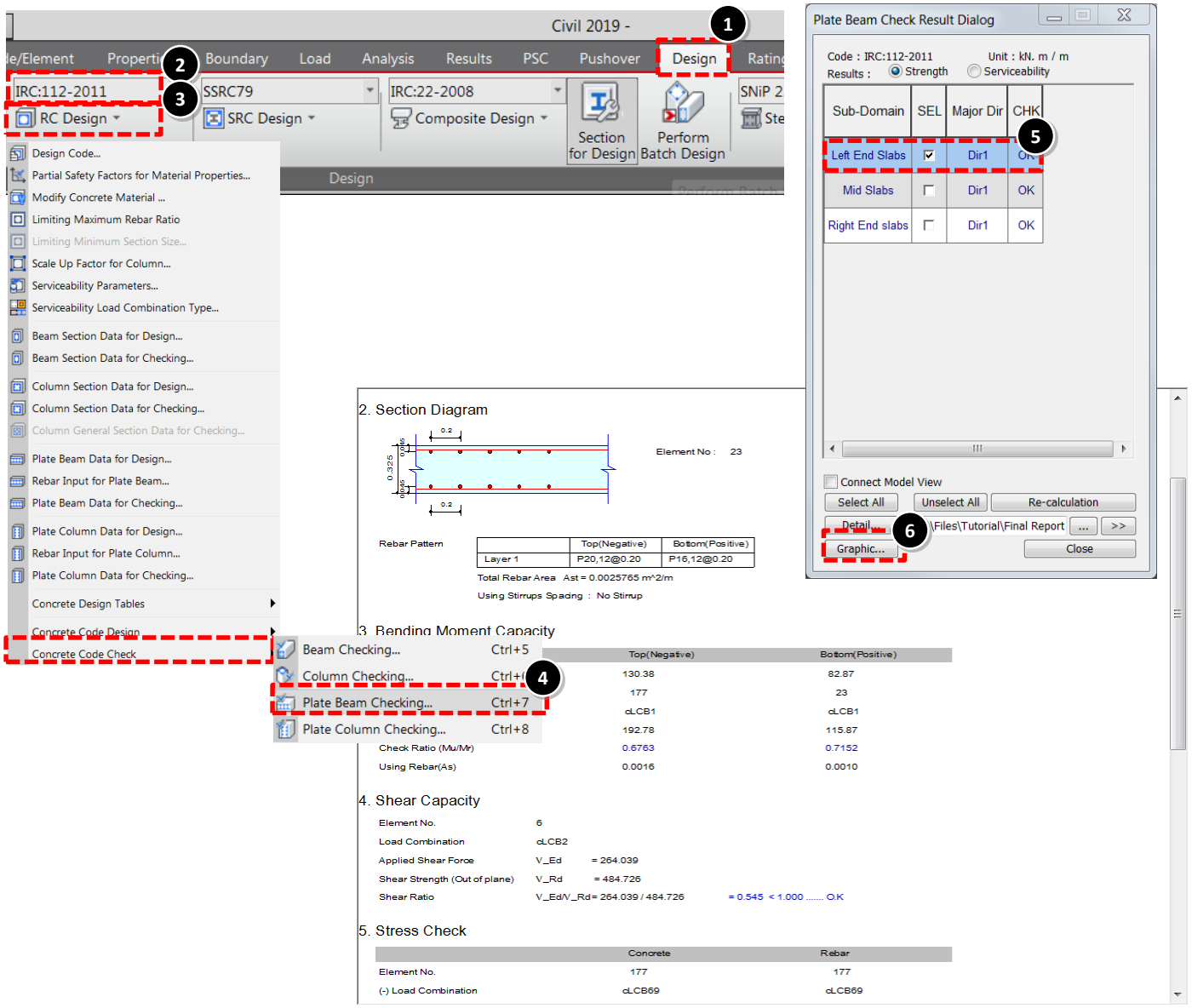




Design – Plate Beam Checking

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**



Design – Plate Column Checking

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**