

LICENCIATURA EN ARQUITECTURA

“ANALISIS DE ESTRUCTURAS”

EXAMEN

Presenta:

KARLA JUDITH ESCOBAR RODRIGUEZ

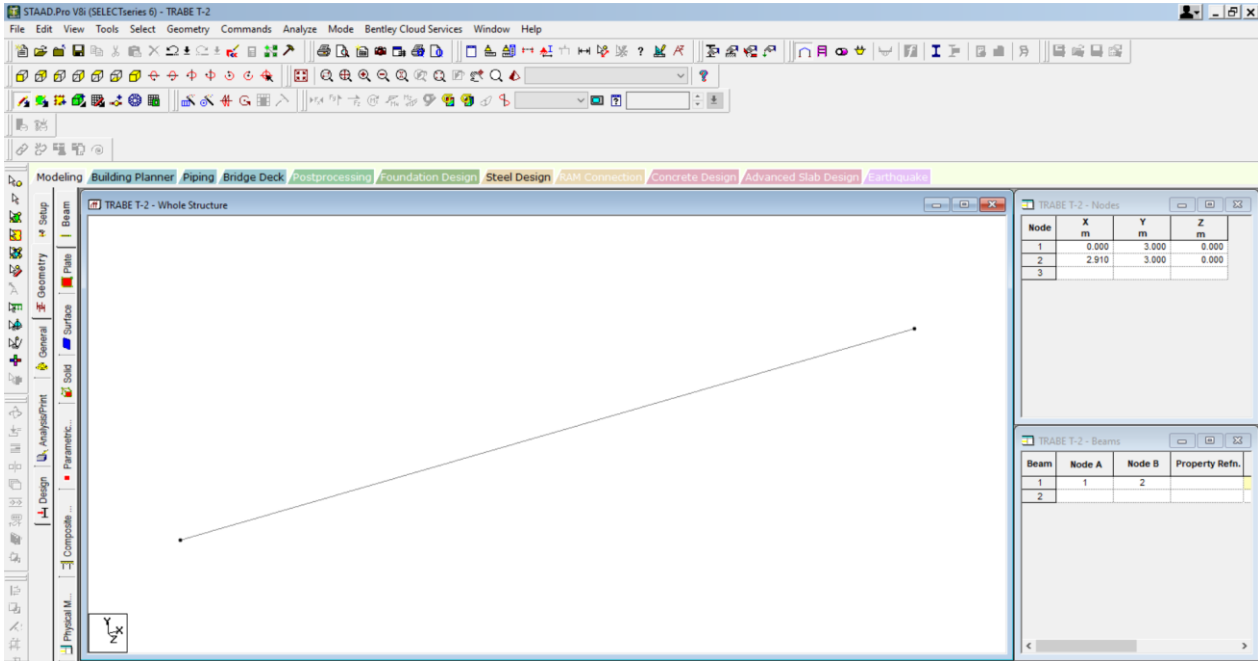
Profesor:

ARQUITECTO. Pedro Alberto García López

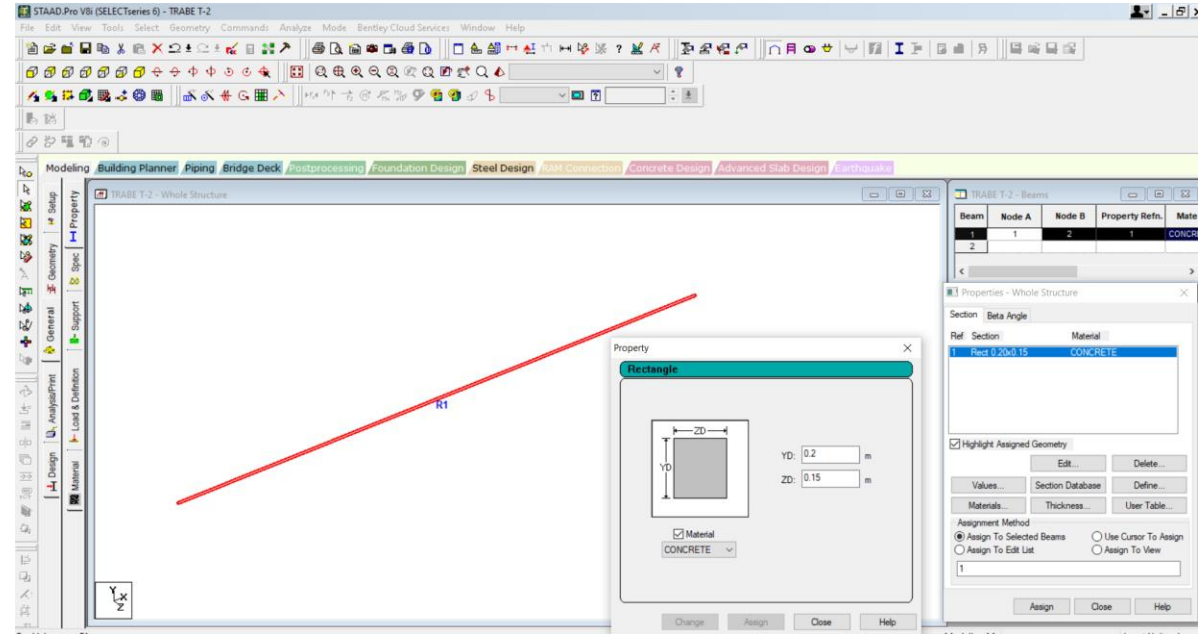


TRABE T3

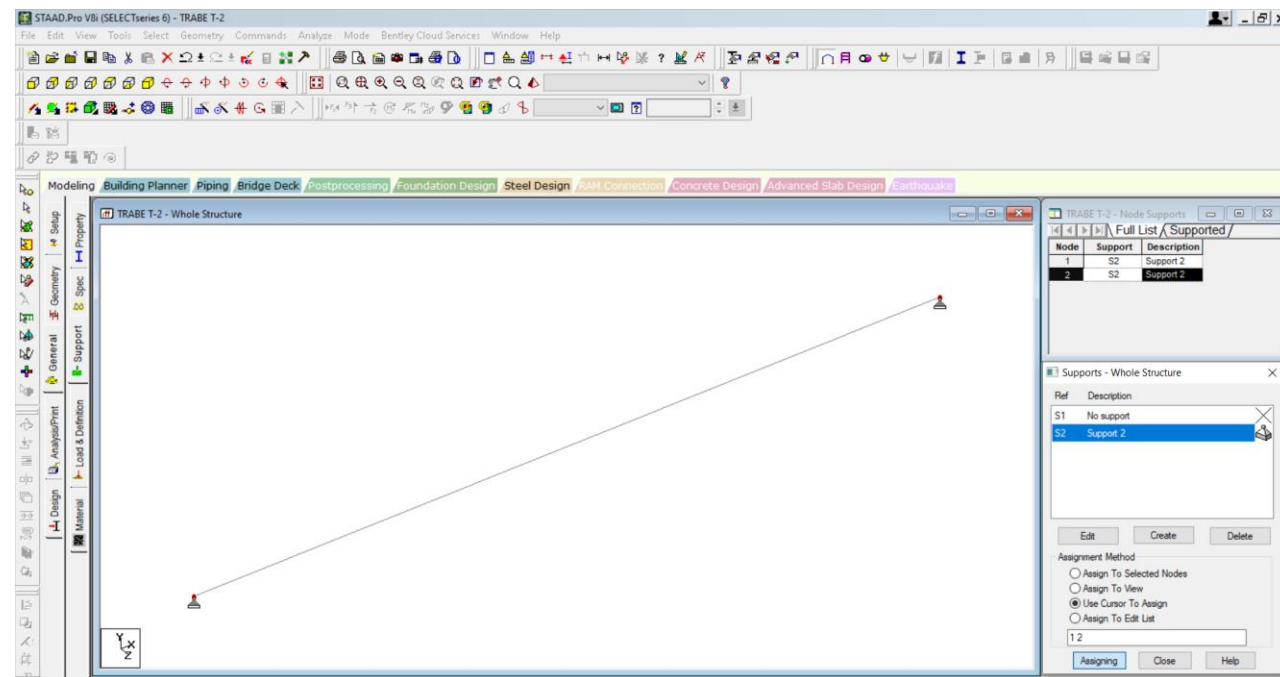
GEOMETRY



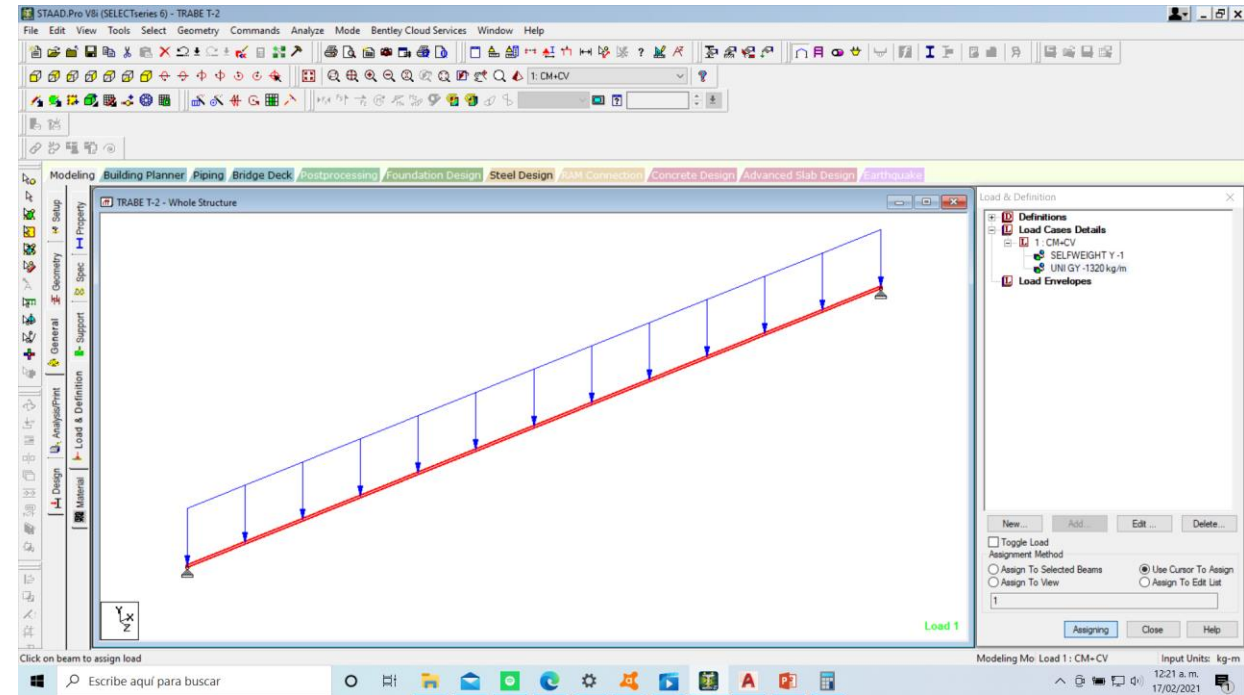
GENERAL



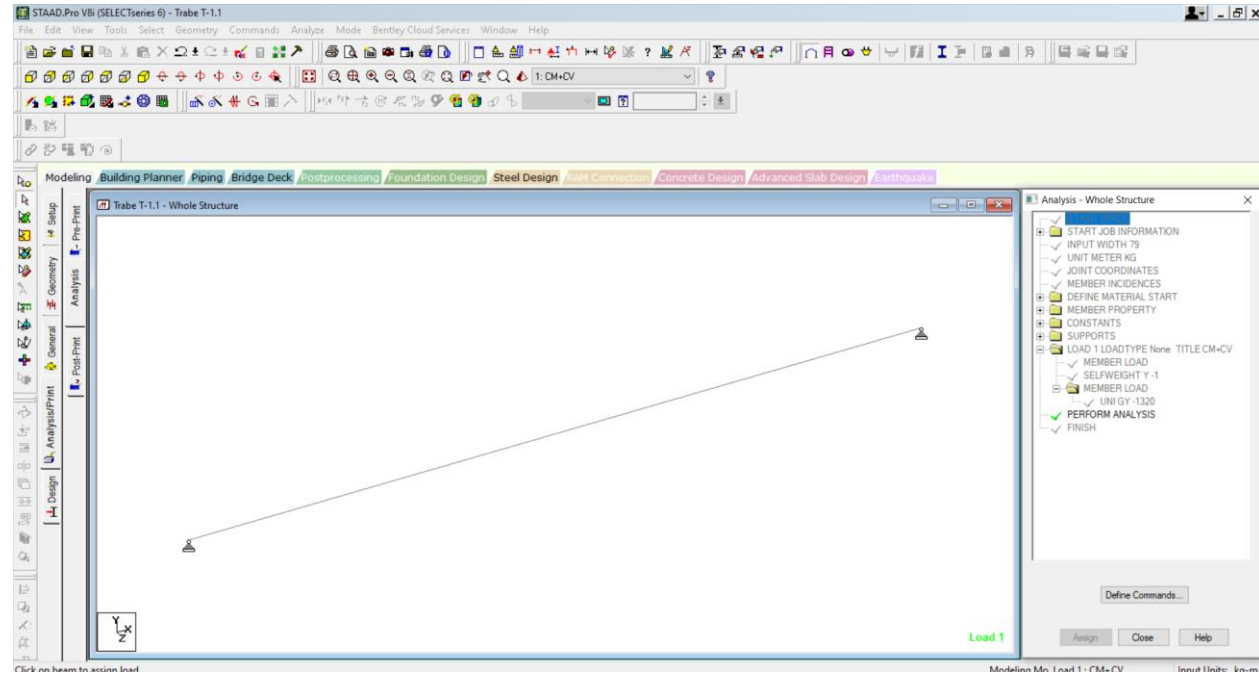
SUPPORT



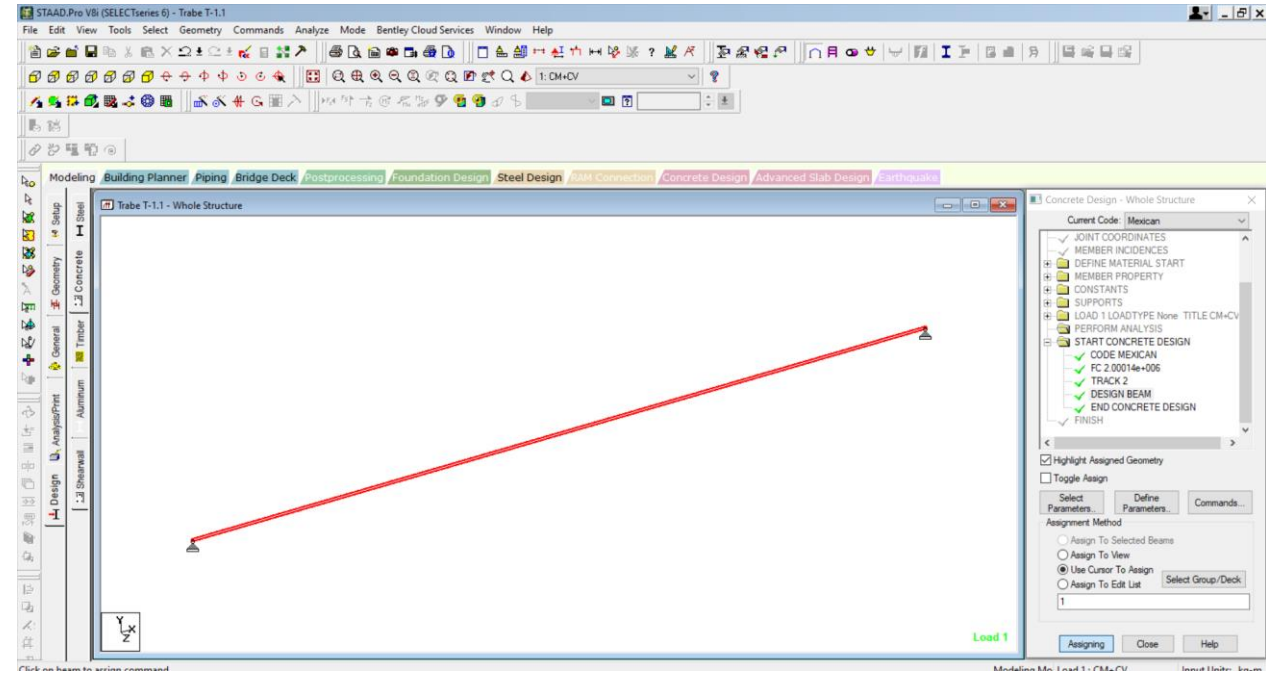
LOAD Y DEFINITION

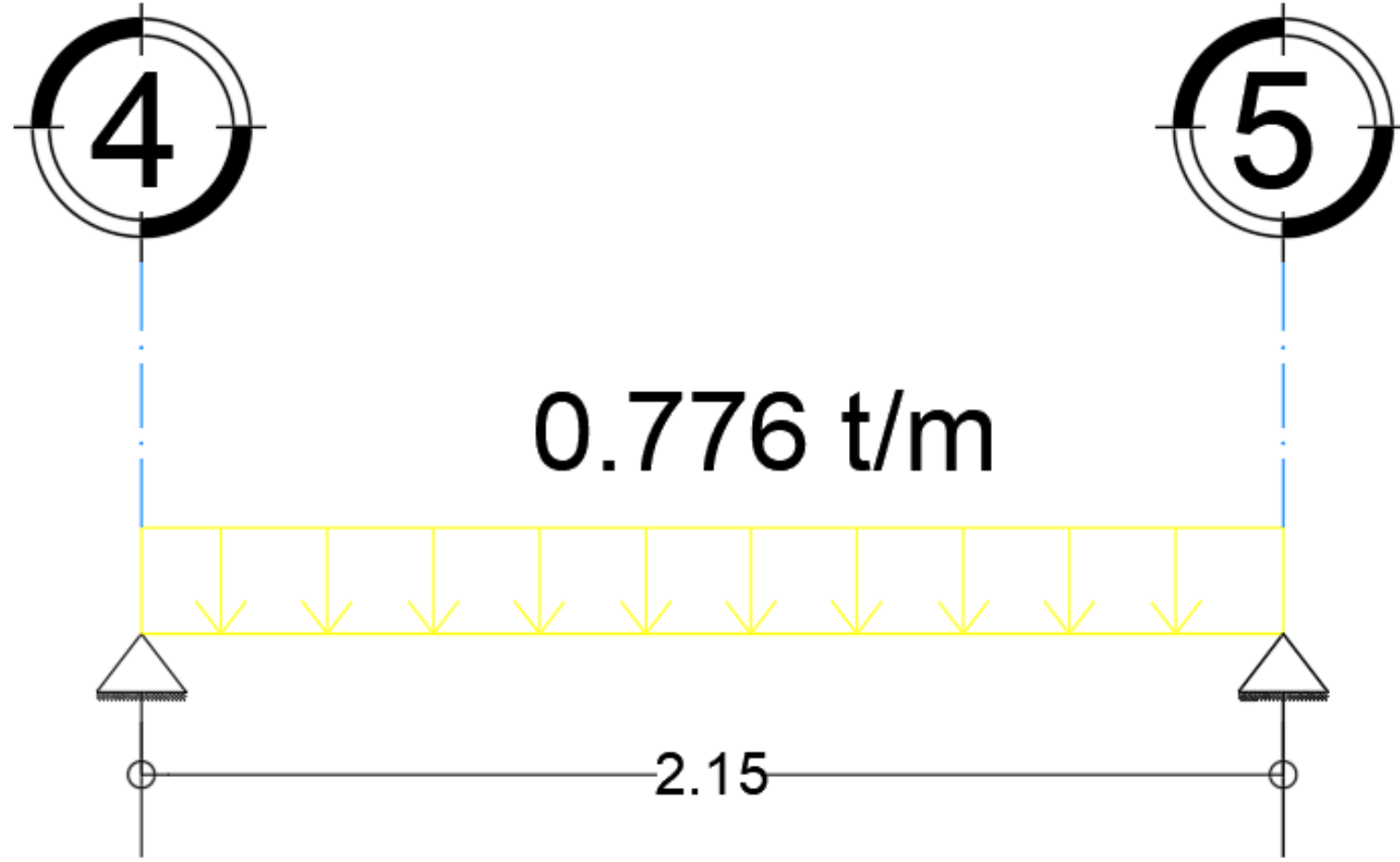


ANALYSIS/PRINT



DESIGN





TRABE T5

GEOMETRY

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) software interface. The main window shows a 3D model of a beam structure. The beam is represented by a line connecting two nodes. The nodes are located at coordinates (0.000, 3.000, 0.000) and (2.150, 3.000, 0.000). The beam is labeled as Beam 1 and Beam 2.

The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help) and a toolbar with various icons for file operations, editing, and analysis. The main window is titled "Trabe T-1.2 - Whole Structure".

On the right side, there are two data tables:

| Node | X m | Y m | Z m |
|------|-------|-------|-------|
| 1 | 0.000 | 3.000 | 0.000 |
| 2 | 2.150 | 3.000 | 0.000 |
| 3 | | | |

| Beam | Node A | Node B | Property Refn. |
|------|--------|--------|----------------|
| 1 | 1 | 2 | |
| 2 | | | |

At the bottom of the interface, there is a status bar with the text "For Help, press F1" on the left, "Modeling Mo" in the center, and "Input Units: kg-m" on the right.

GENERAL

STAAD.Pro V8i (SELECTseries 6) - Trabe T-1.2

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

Trabe T-1.2 - Whole Structure

Trabe T-1.2 - Beams

| Beam | Node A | Node B | Property Refn. | Mate |
|------|--------|--------|----------------|-------|
| 1 | 1 | 2 | 1 | CONCR |
| 2 | | | | |

Properties - Whole Structure

Section Beta Angle

| Ref | Section | Material |
|-----|----------------|----------|
| 1 | Rect 0.20x0.15 | CONCRETE |

Highlight Assigned Geometry

Values... Section Database Define...
Materials... Thickness... User Table...

Assignment Method

Assign To Selected Beams Use Cursor To Assign
 Assign To Edit List Assign To View

1

Assign Close Help

Property

Rectangle

YD: 0.2 m
ZD: 0.15 m

Material
CONCRETE

Change Assign Close Help

For Help, press F1

Modeling Mo Input Units: kg-m

SUPPORT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) software interface. The main window shows a 2D model of a structure with two support nodes, each represented by a blue pin icon. A diagonal line connects the two nodes. The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar with various icons, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The left sidebar contains a vertical toolbar with icons for Setup, Property, Spec, Support, General, Geometry, Analysis/Print, Load & Definition, Design, and Material.

The 'trabe t-3 - Node Supports' dialog box is open, showing a table with the following data:

| Node | Support | Description |
|------|---------|-------------|
| 1 | S2 | Support 2 |
| 2 | S2 | Support 2 |

The 'Supports - Whole Structure' dialog box is also open, showing a table with the following data:

| Ref | Description |
|-----|-------------|
| S1 | No support |
| S2 | Support 2 |

The 'Assignment Method' section in the 'Supports - Whole Structure' dialog box has the following options:

- Assign To Selected Nodes
- Assign To View
- Use Cursor To Assign
- Assign To Edit List

The 'Assigning' button is highlighted.

LOAD Y DEFINITION

The screenshot displays the STAAD.Pro V8i (SELECTSeries 6) - trabe t-3 interface. The main window shows a 3D model of a trapezoidal structure with a distributed load applied. The 'Load & Definition' dialog box is open, showing the load case '1: CM+CV' and its components: 'SELFWEIGHT Y -1' and 'UNI GY -776 kg/m'. The dialog also includes options for 'Toggle Load' and 'Assignment Method'.

Click on beam to assign load

Modeling Mo. Load 1: CM+CV Input Units: kg-m

ANALYSIS/PRINT

The screenshot displays the STAAD.Pro V8i (SELECTSeries 6) software interface. The main window is titled "trabe t-3 - Whole Structure" and shows a simple structural model consisting of a single diagonal member with a support at the bottom left and a load at the top right. The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar with various icons, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The "Analysis/Print" tab is active, and the "Analysis" sub-tab is selected. A vertical toolbar on the left contains icons for Setup, Geometry, Analysis, General, Post-Print, Design, and Analysis/Print. The "Analysis - Whole Structure" panel on the right shows a tree view of the analysis process, including steps like START JOB INFORMATION, INPUT WIDTH 79, UNIT METER KG, JOINT COORDINATES, MEMBER INCIDENCES, DEFINE MATERIAL START, MEMBER PROPERTY, CONSTANTS, SUPPORTS, LOAD 1 LOADTYPE None TITLE CM+CV, SELFWEIGHT Y -1, MEMBER LOAD, UNI GY -776, PERFORM ANALYSIS, and FINISH. The "PERFORM ANALYSIS" step is highlighted with a green checkmark. At the bottom of the main window, there is a status bar with the text "Click on beam to assign load" on the left, "Modeling Mo Load 1 : CM+CV" in the center, and "Input Units: kg-m" on the right.

STAAD.Pro V8i (SELECTSeries 6) - trabe t-3

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

trabe t-3 - Whole Structure

Analysis - Whole Structure

- START JOB INFORMATION
- INPUT WIDTH 79
- UNIT METER KG
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CM+CV
 - SELFWEIGHT Y -1
 - MEMBER LOAD
 - UNI GY -776
- PERFORM ANALYSIS
- FINISH

Define Commands...

Assign Close Help

Click on beam to assign load

Modeling Mo Load 1 : CM+CV

Input Units: kg-m

DESIGN

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) software interface. The main window shows a 3D model of a structure named "trabe t-3 - Whole Structure" with a single diagonal member. The software is running in the "Concrete Design" module, as indicated by the highlighted tab in the top ribbon. The right-hand panel, titled "Concrete Design - Whole Structure", shows the design progress for the current code "Mexican". The design steps are listed as follows:

- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CM+CV
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Below the list, the "Highlight Assigned Geometry" checkbox is checked, and the "Toggle Assign" checkbox is unchecked. The "Assignment Method" section shows "Use Cursor To Assign" selected. The "Assign" button is visible at the bottom of the panel. The status bar at the bottom indicates "Modeling Mode Load 1: CM+CV" and "Input Units: kn-m".

ANALISE

STAAD.Pro V8i (SELECTseries 6) - trabe t-3

trabe t-3.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

BEAM NO. 1 DESIGN RESULTS - FLEXURE

PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES, DDF

LEN - 2150.00 (mm) FY - 412. FC - 20. SIZE - 150.00 X 200.00 (mm)

| LEVEL | HEIGHT (mm) | BAR INFO | FROM (mm) | TO (mm) | ANCHOR STA | ANCHOR END |
|-------|-------------|----------|-----------|---------|------------|------------|
| 1 | 42. | 2 - 2.MM | 0. | 2150. | YES | YES |

CRITICAL POS MOMENT= 4.81 kNm AT 1075.00 (mm) LOAD 1

REQD STEEL= 86.91 (mm²) ROW=0.0037 ROWMX=0.0152 ROWMN=0.0018

REQD COMP STEEL= 0.00 (mm²)

MAX/MIN/ACTUAL BAR SPACING= 66.22/ 37.90/ 66.22 (mm)

COMP MAX/MIN/ACTUAL BAR SPACING= 0.00/ 0.00/ 0.00 (mm)

BASIC/REQD. DEVELOPMENT LENGTH = 199.08/ 266.04 (mm)

Cracked Moment of Inertia I_z at above location = 0.13657E+08 mm⁴

REQUIRED REINF. STEEL SUMMARY :

| SECTION | REINF STEEL (+VE/-VE) | MOMENTS (+VE/-VE) | LOAD (+VE/-VE) |
|---------|-----------------------|-------------------|----------------|
| | | | |

Notes: NOTES, RESULTS

Total Page: 4 CAP NUM

Concrete Design - Whole Structure

Current Code: Mexican

- UNIT METER KG
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CM+CV
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Highlight Assigned Geometry

Toggle Assign

Select Parameters... Define Parameters... Commands...

Assignment Method

Assign To Selected Beams

Assign To View

Use Cursor To Assign

Assign To Edit List

Select Group/Deck

1

Assign Close Help

Load 1

Modeling Mo: Load 1: CM+CV Input Units: kg-m

For Help, press F1

ANALISE

trabe t-3.anl - STAAD Output Viewer

File Edit View Help

***WARNING - INSTABILITY AT JOINT

REQUIRED REINF. STEEL SUMMARY :

| SECTION (MM) | REINF STEEL (+VE/-VE) (SQ. MM) | MOMENTS (+VE/-VE) (KNS-MET) | LOAD (+VE/-VE) |
|-------------------|------------------------------------|---------------------------------|----------------|
| 0.00 | 0.00/ | 0.00 | 0/ 1 |
| 179.17 | 39.29/ | 0.00 | 1/ 0 |
| 358.33 | 52.33/ | 0.00 | 1/ 0 |
| 537.50 | 71.68/ | 0.00 | 1/ 0 |
| 716.67 | 85.88/ | 0.00 | 1/ 0 |
| 895.83 | 94.57/ | 0.00 | 1/ 0 |
| 1075.00 | 97.49/ | 0.00 | 1/ 0 |
| 1254.17 | 94.57/ | 0.00 | 1/ 0 |
| 1433.33 | 85.88/ | 0.00 | 1/ 0 |
| 1612.50 | 71.68/ | 0.00 | 1/ 0 |
| 1791.67 | 52.33/ | 0.00 | 1/ 0 |
| 1970.83 | 50.56/ | 0.00 | 1/ 0 |
| 2150.00 | 0.00/ | 0.00 | 0/ 1 |

BEAM NO. 1 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu= 0.02 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1

STIRRUPS ARE NOT REQUIRED.

NOTES

RESULTS

Total Page: 4 CAP NUM

ANALISE

trabe t-3.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

BEAM NO. 1 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu= 0.02 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1
STIRRUPS ARE NOT REQUIRED.

-----< PAGE 3 Ends Here >-----

STAAD SPACE -- PAGE NO. 4

AT END SUPPORT - Vu= 0.02 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1
STIRRUPS ARE NOT REQUIRED.

37. END CONCRETE DESIGN
38. FINISH

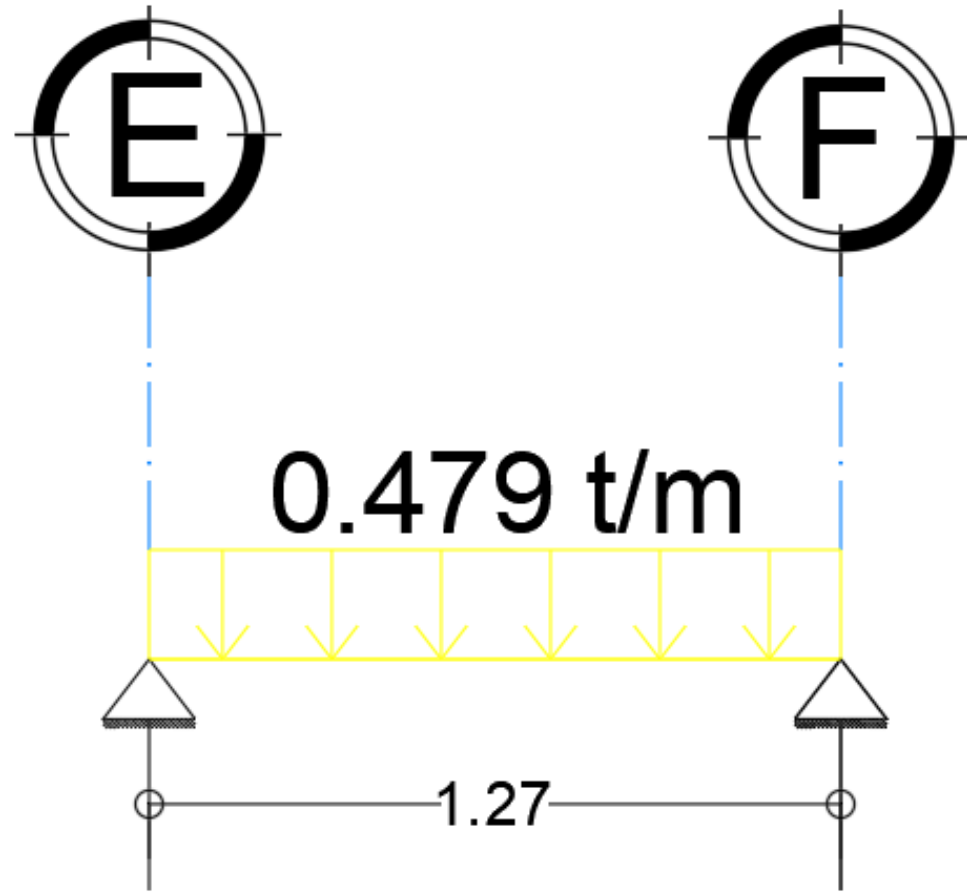
***** END OF THE STAAD.Pro RUN *****

**** DATE= FEB 17,2021 TIME= 2: 8: 2 ****

* For technical assistance on STAAD.Pro, please visit *
* <http://selectservices.bentley.com/en-US/> *
* *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *

NOTES
RESULTS

Total Page: 4 CAP NUM



TRABE T9

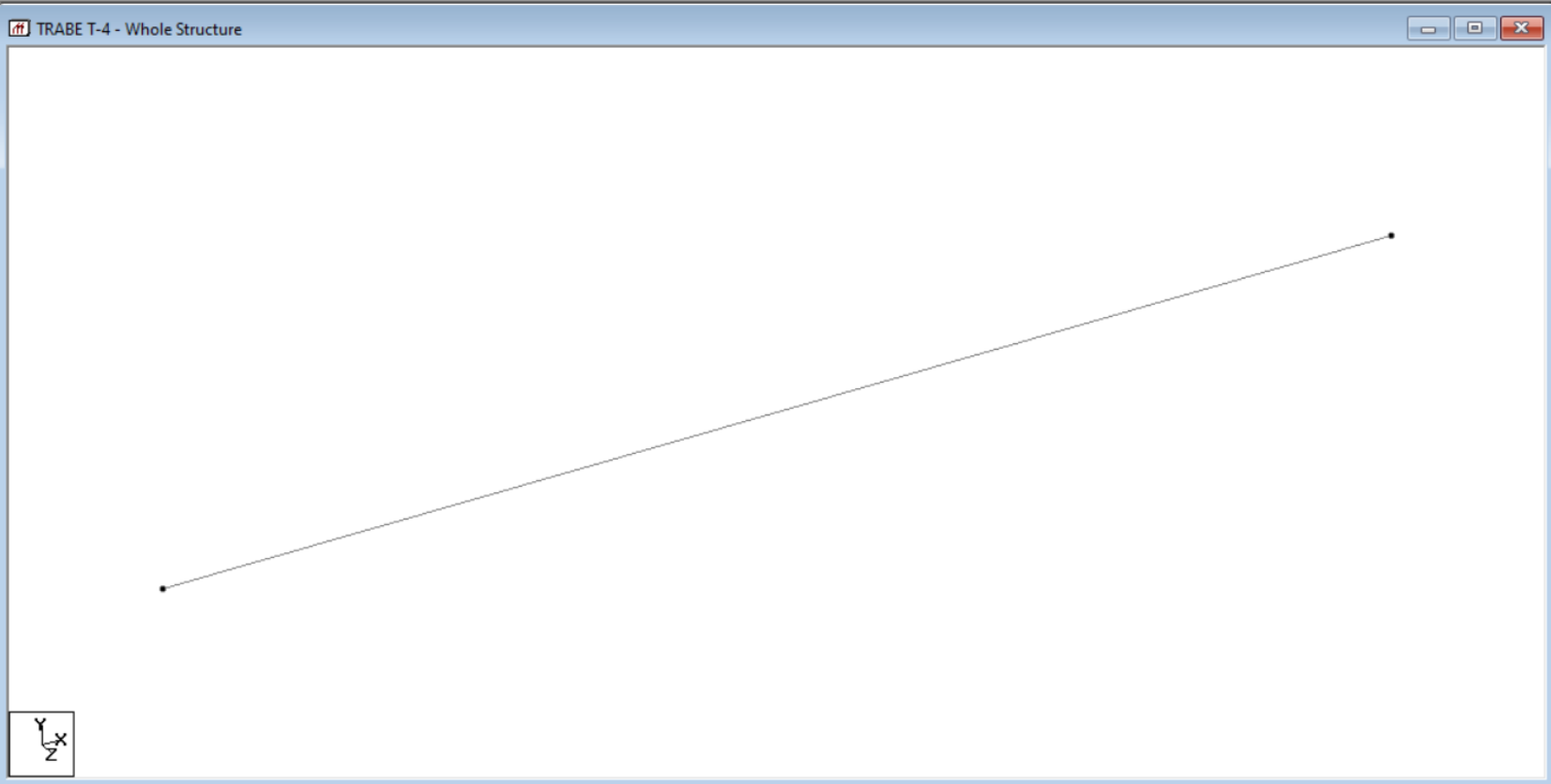
GEOMETRY

STAAD.Pro V8i (SELECTseries 6) - TRABE T-4

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

TRABE T-4 - Whole Structure



TRABE T-4 - Nodes

| Node | X m | Y m | Z m |
|------|--------|--------|--------|
| 1 | 0.000 | 3.000 | 0.000 |
| 2 | 1.260 | 3.000 | 0.000 |
| 3 | | | |

TRABE T-4 - Beams

| Beam | Node A | Node B | Property Refn. |
|------|--------|--------|----------------|
| 1 | 1 | 2 | |
| 2 | | | |

Click on node at start of beam

Modeling Mo

Input Units: kg-m

GENERAL

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 3D model of a beam labeled 'R1' in blue. A red double-line beam is visible in the model. A 'Property' dialog box is open, showing a 'Rectangle' section with dimensions YD: 0.1 m and ZD: 0.15 m. The material is set to 'CONCRETE'. The 'Assignment Method' is set to 'Assign To Selected Beams'. The 'Properties - Whole Structure' window is also open, showing a table of beam properties.

| Beam | Node A | Node B | Property Refn. | Mate |
|------|--------|--------|----------------|-------|
| 1 | 1 | 2 | 1 | CONCR |
| 2 | | | | |

| Ref | Section | Material |
|-----|----------------|----------|
| 1 | Rect 0.10x0.15 | CONCRETE |

For Help, press F1

Modeling Mo

Input Units: kq-m

SUPPORT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 2D model of a beam with two supports. The 'Supports - Whole Structure' dialog box is open, showing a table of supports and assignment options.

Supports - Whole Structure

| Ref | Description |
|-----|-------------|
| S1 | No support |
| S2 | Support 2 |

Assignment Method:

- Assign To Selected Nodes
- Assign To View
- Use Cursor To Assign
- Assign To Edit List

12

Assigning Close Help

Click on node to set support

Modeling Mo

Input Units: kg-m

LOAD Y DEFINITION

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 3D model of a beam structure with a distributed load applied along its length. The load is represented by a series of downward-pointing blue arrows. The beam is supported by a pin support at the left end and a roller support at the right end. The software interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar with various icons, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The 'Load & Definition' dialog box is open on the right side, showing a tree view of the load definitions. The tree view includes 'Definitions', 'Load Cases Details', and 'Load Envelopes'. Under 'Load Cases Details', there is a sub-entry '1 : CV+CM' which contains 'SELFWEIGHT Y -1' and 'UNI GY -479 kg/m'. The dialog box also has buttons for 'New...', 'Add...', 'Edit...', and 'Delete...'. Below these buttons, there are options for 'Toggle Load' and 'Assignment Method'. The 'Assignment Method' section has three radio buttons: 'Assign To Selected Beams/Plates', 'Use Cursor To Assign' (which is selected), and 'Assign To Edit List'. There is also a text input field containing the number '1'. At the bottom of the dialog box, there are buttons for 'Assign', 'Close', and 'Help'. The status bar at the bottom of the software window shows 'Modeling Mo Load 1 : CV+CM' and 'Input Units: kg-m'.

STAAD.Pro V8i (SELECTseries 6) - TRABE T-4

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

TRABE T-4 - Whole Structure

Load & Definition

- Definitions
- Load Cases Details
 - 1 : CV+CM
 - SELFWEIGHT Y -1
 - UNI GY -479 kg/m
- Load Envelopes

New... Add... Edit... Delete...

Toggle Load

Assignment Method

Assign To Selected Beams/Plates Use Cursor To Assign

Assign To View Assign To Edit List

1

Assign Close Help

For Help, press F1

Modeling Mo Load 1 : CV+CM Input Units: kg-m

ANALYSIS/PRINT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 2D model of a beam with two pin supports, labeled "Load 1". The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar with various icons, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The "Analysis/Print" tab is active, showing a tree view of the analysis process. The tree view includes:

- START JOB INFORMATION
- INPUT WIDTH 79
- UNIT METER KG
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CV+CM
 - SELFWEIGHT Y -1
 - MEMBER LOAD
 - UNI GY -479
- PERFORM ANALYSIS
- FINISH

The status bar at the bottom indicates "Modeling Mo Load 1 : CV+CM" and "Input Units: kg-m".

DESIGN

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 3D model of a structure with a single member highlighted in green, labeled "Load 1". The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help) and a toolbar with various icons for modeling and analysis. The "Concrete Design" tab is active in the top navigation bar.

The "Concrete Design - Whole Structure" dialog box is open on the right side, showing the following details:

- Current Code: Mexican
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CV+CM
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Additional options in the dialog include:

- Highlight Assigned Geometry
- Toggle Assign
- Buttons: Select Parameters..., Define Parameters..., Commands...
- Assignment Method:
 - Assign To Selected Beams
 - Assign To View
 - Use Cursor To Assign
 - Assign To Edit List
- Input field: 1
- Buttons: Assign, Close, Help

At the bottom of the interface, the status bar displays: "For Help, press F1", "Modeling Mo: Load 1 : CV+CM", and "Input Units: kg-m".

ANALISE

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-4 software interface. The main window shows a 3D model of a structure with a load applied. A 'STAAD Analysis and Design' dialog box is open, displaying the following log:

```
Current Directory: C:\SProV8i SS6\STAAD
Input File: TRABE T-4.std

++ Processing Joint Coordinates.          2:25:59
++ Processing Member Information.         2:25:59
++ Reading Member Properties ...          2:25:59
++ Finished Reading Member Properties ... 0 ms
++ Processing Support Condition.          2:25:59
++ Read/Check Data in Load Cases ...     2:25:59
++ Using In-Core Advanced Math Solver
++ Processing and setting up Load Vector. 2:25:59
++ Processing Element Stiffness Matrix.   2:25:59
++ Calculating Member Forces.             2:25:59
++ Analysis Successfully Completed ++
++ Performing Concrete Design             2:25:59
++ Calculating Section Forces1            2:25:59
++ Calculating Section Forces2            2:25:59
++ Start Concrete Design ...              2:25:59
++ Creating Displacement File (DGP)...     2:26: 7
++ Creating Reaction File (REA)...         2:26: 7
++ Calculating Section Forces1-110.       2:26: 7
++ Calculating Section Forces2.           2:26: 7
++ Creating Section Force File (BMD)...    2:26: 7
++ Creating Section Displace File (SCH)... 2:26: 7
++ Creating Design information File (DGN)... 2:26: 7
++ Done.                                  2:26: 7

0 Error(s). 1 Warning(s). 1 Note(s)
++ End STAAD.Pro Run Elapsed Time = 8 Secs
C:\SProV8i SS6\STAAD\TRABE T-4.anl
```

Below the log, there are three radio buttons: 'View Output File', 'Go to Post Processing Mode', and 'Stay in Modeling Mode' (selected). A 'Done' button is at the bottom right of the dialog.

The 'Concrete Design - Whole Structure' dialog box is also open, showing the following settings:

- Current Code: Mexican
- UNIT METER KG
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CV+CM
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Additional options include 'Highlight Assigned Geometry' (checked), 'Toggle Assign' (unchecked), and 'Assignment Method' (Use Cursor To Assign selected). A 'Select Group/Deck' button and a text box containing '1' are also visible.

The bottom status bar shows 'Modeling Mo Load 1: CV+CM' and 'Input Units: kg-m'.

ANALISE

TRABE T-4.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

BEAM NO. 1 DESIGN RESULTS - FLEXURE

PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES,DDF

LEN - 1260.00 (mm) FY - 412. FC - 20. SIZE - 150.00 X 100.00 (mm)

| LEVEL | HEIGHT (mm) | BAR INFO | FROM (mm) | TO (mm) | ANCHOR STA END | |
|-------|----------------|----------|--------------|------------|-------------------|-----|
| 1 | 42. | 2 - 2.MM | 0. | 1260. | YES | YES |

-----|
| CRITICAL POS MOMENT= 1.00 kNm AT 630.00 (mm) LOAD 1|
| REQD STEEL= 51.17 (mm2) ROW=0.0059 ROWMX=0.0152 ROWMN=0.0033 |
| REQD COMP STEEL= 0.00 (mm2) |
| MAX/MIN/ACTUAL BAR SPACING= 66.22/ 37.90/ 66.22 (mm) |
| COMP MAX/MIN/ACTUAL BAR SPACING= 0.00/ 0.00/ 0.00 (mm) |
| BASIC/REQD. DEVELOPMENT LENGTH = 199.08/ 178.30 (mm) |
|-----|

Cracked Moment of Inertia I_z at above location =0.91086E+06 mm⁴

REQUIRED REINF. STEEL SUMMARY :

| SECTION | REINF STEEL (+VE/-VE) | MOMENTS (+VE/-VE) | LOAD (+VE/-VE) |
|---------|-----------------------|-------------------|----------------|
|---------|-----------------------|-------------------|----------------|

NOTES

RESULTS

ANALISE

TRABE T-4.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

Cracked Moment of Inertia I_z at above location =0.91086E+06 mm⁴

REQUIRED REINF. STEEL SUMMARY :

| SECTION (MM) | REINF STEEL (+VE/-VE) (SQ. MM) | MOMENTS (+VE/-VE) (KNS-MET) | LOAD (+VE/-VE) |
|-------------------|------------------------------------|---------------------------------|----------------|
| 0.00 | 0.00/ 0.00 | 0./ 0.00 | 0/ 1 |
| 105.00 | 21.43/ 0.00 | 0./ 0.00 | 1/ 0 |
| 210.00 | 38.47/ 0.00 | 1./ 0.00 | 1/ 0 |
| 315.00 | 54.17/ 0.00 | 1./ 0.00 | 1/ 0 |
| 420.00 | 66.44/ 0.00 | 1./ 0.00 | 1/ 0 |
| 525.00 | 74.35/ 0.00 | 1./ 0.00 | 1/ 0 |
| 630.00 | 77.08/ 0.00 | 1./ 0.00 | 1/ 0 |
| 735.00 | 74.35/ 0.00 | 1./ 0.00 | 1/ 0 |
| 840.00 | 66.44/ 0.00 | 1./ 0.00 | 1/ 0 |
| 945.00 | 54.17/ 0.00 | 1./ 0.00 | 1/ 0 |
| 1050.00 | 38.47/ 0.00 | 1./ 0.00 | 1/ 0 |
| 1155.00 | 21.43/ 0.00 | 0./ 0.00 | 1/ 0 |
| 1260.00 | 0.00/ 0.00 | 0./ 0.00 | 0/ 1 |

BEAM NO. 1 DESIGN RESULTS - SHEAR

NOTES

RESULTS

Total Page: 4 CAP NUM

ANALISE

TRABE T-4.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

```

      B E A M N O .      1 D E S I G N R E S U L T S - S H E A R

      AT START SUPPORT - Vu=   0.01 KN Vc=   0.00 KN Vs=   0.00 KN
Tu=   0.00 Kn Me Tc=   0.00 Kn Me Ts=   0.00 Kn Me LOAD   1
                        STIRRUPS ARE NOT REQUIRED.

-----< PAGE 3 Ends Here >-----
      S T A A D   S P A C E                                -- PAGE NO.   4

      AT END   SUPPORT - Vu=   0.01 KN Vc=   0.00 KN Vs=   0.00 KN
Tu=   0.00 Kn Me Tc=   0.00 Kn Me Ts=   0.00 Kn Me LOAD   1
                        STIRRUPS ARE NOT REQUIRED.

      37. END CONCRETE DESIGN
      38. FINISH

      ***** END OF THE STAAD.Pro RUN *****

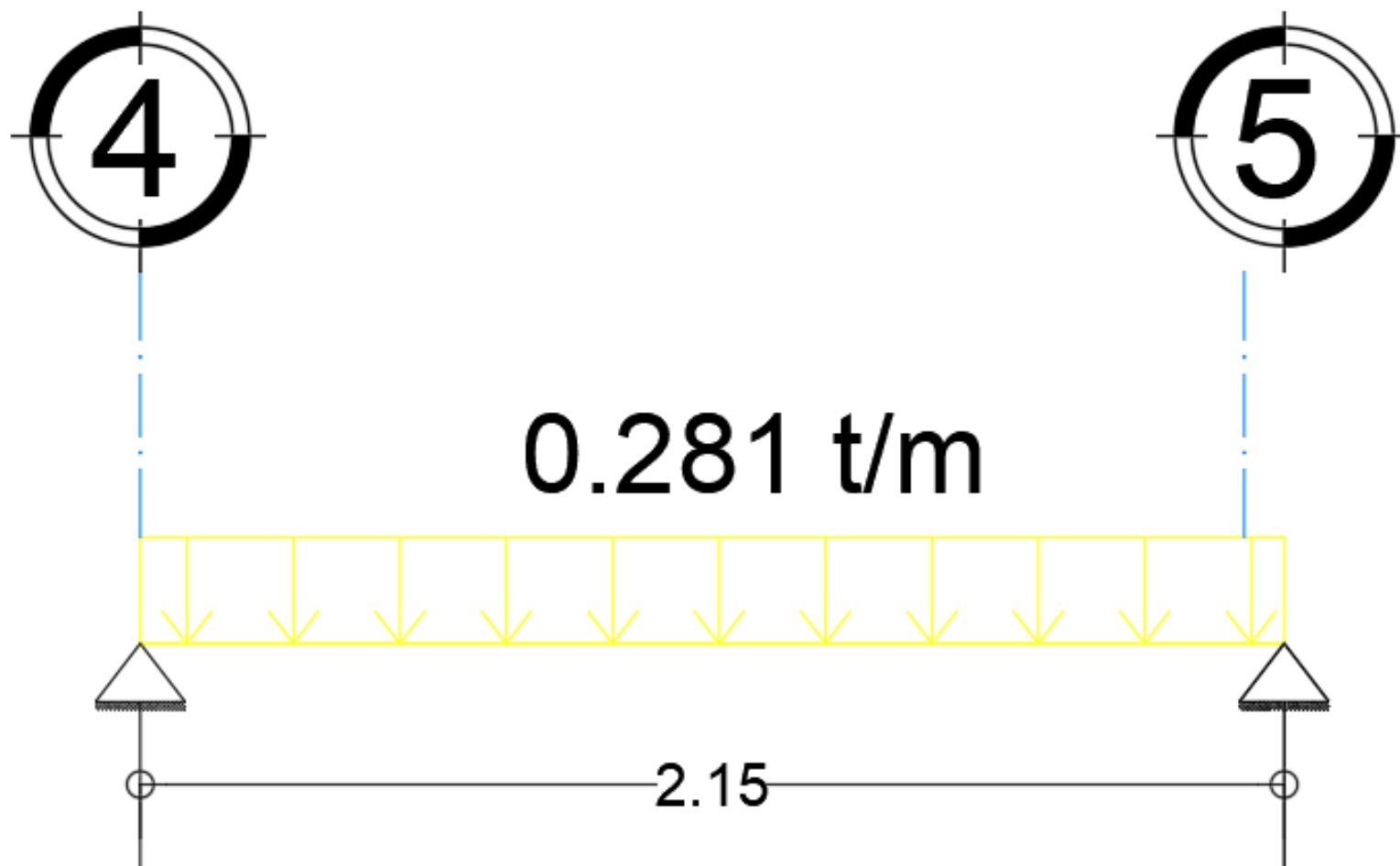
      **** DATE= FEB 17,2021   TIME=  2:26: 7 ****

      *****
      *   For technical assistance on STAAD.Pro, please visit   *
      *   http://selectservices.bentley.com/en-US/              *
      *                                                         *
      *   Details about additional assistance from              *
      *   Bentley and Partners can be found at program menu    *
      *   Help->Technical Support                               *
      *****

      Total Page : 4 CAP NIUM
```

NOTES

RESULTS



TRABE T6

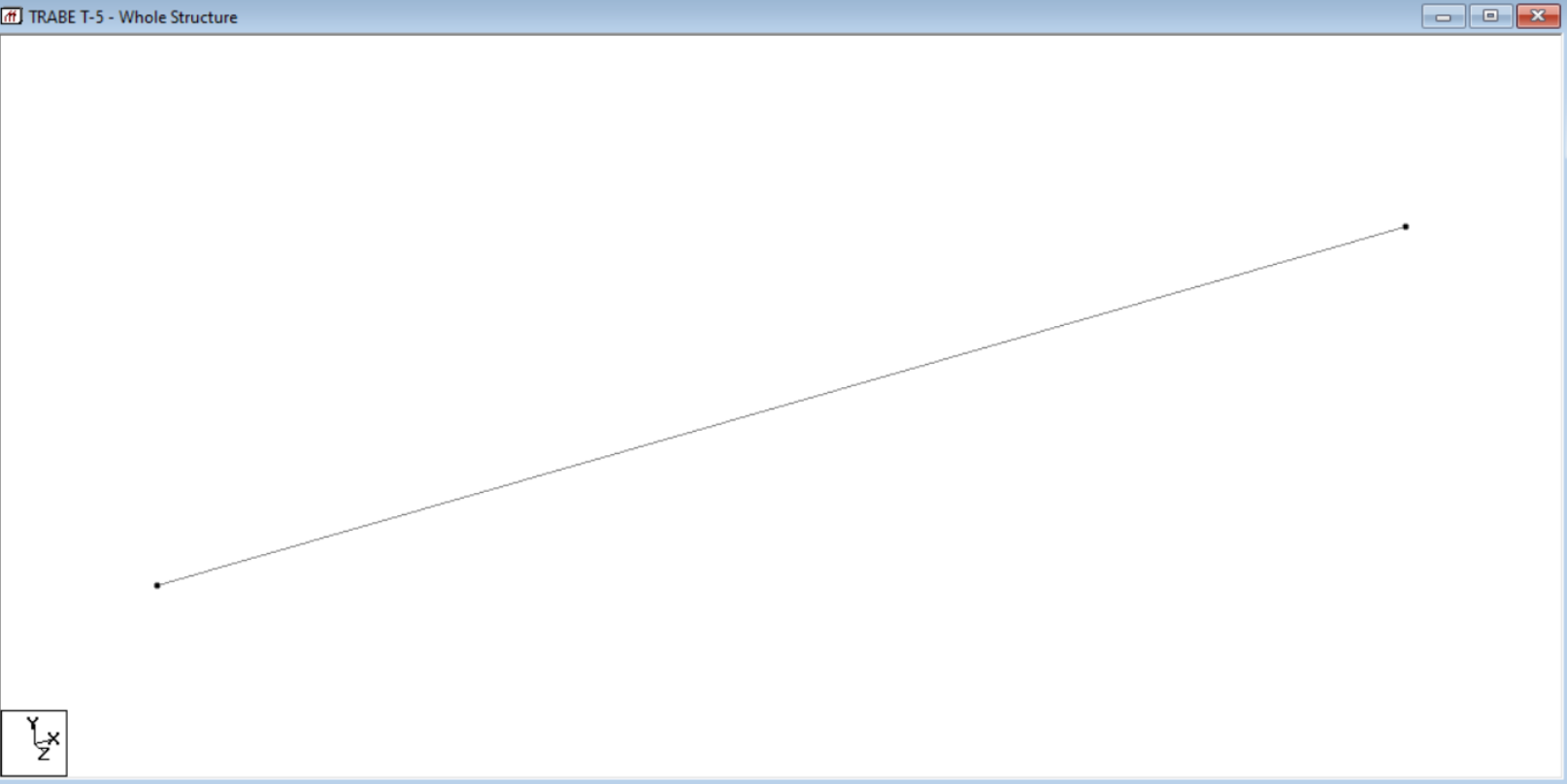
GEOMETRY

STAAD.Pro V8i (SELECTseries 6) - TRABE T-5

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

TRABE T-5 - Whole Structure



TRABE T-5 - Nodes

| Node | X m | Y m | Z m |
|------|-------|-------|-------|
| 1 | 0.000 | 3.000 | 0.000 |
| 2 | 2.150 | 3.000 | 0.000 |
| 3 | | | |

TRABE T-5 - Beams

| Beam | Node A | Node B | Property Refn. |
|------|--------|--------|----------------|
| 1 | 1 | 2 | |
| 2 | | | |

Click on node at start of beam

Modeling Mo

Input Units: kg-m

GENERAL

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-5 software interface. The main window shows a 3D view of a structure with a red beam element labeled R1. A coordinate system (Y, Z, X) is visible in the bottom left corner of the main window.

The **Property** dialog box is open, showing the **Rectangle** section. The dimensions are YD: 0.2 m and ZD: 0.15 m. The material is set to CONCRETE.

The **Properties - Whole Structure** dialog box is also open, showing the **Section** and **Beta Angle** tabs. The section is **Rect 0.20x0.15** and the material is **CONCRETE**. The assignment method is set to **Assign To Selected Beams**.

| Beam | Node A | Node B | Property Refn. | Mate |
|------|--------|--------|----------------|-------|
| 1 | 1 | 2 | 1 | CONCR |
| 2 | | | | |

| Ref | Section | Material |
|-----|----------------|----------|
| 1 | Rect 0.20x0.15 | CONCRETE |

For Help, press F1

Modeling Mo

Input Units: kg-m

SUPPORT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-5 software interface. The main window shows a 2D model of a beam with two supports, represented by blue triangles. The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The left sidebar contains various tool icons and a vertical menu with options like Setup, Property, Spec, Support, General, Geometry, Analysis/Print, Design, and Material. Two dialog boxes are open on the right side:

- TRABE T-5 - Node Supports**: A dialog box with a table showing the current support assignments for nodes 1 and 2.
- Supports - Whole Structure**: A dialog box for assigning supports to nodes, with a list of support types (S1: No support, S2: Support 2) and an assignment method section.

| Node | Support | Description |
|------|---------|-------------|
| 1 | S2 | Support 2 |
| 2 | S2 | Support 2 |

| Ref | Description |
|-----|-------------|
| S1 | No support |
| S2 | Support 2 |

Assignment Method:

- Assign To Selected Nodes
- Assign To View
- Use Cursor To Assign
- Assign To Edit List

Input field: 1 2

Buttons: Assigning, Close, Help

Click on node to set support

Modeling Mo

Input Units: kg m

LOAD Y DEFINITION

The screenshot displays the STAAD.Pro V8i (SELECTSeries 6) - TRABE T-5 software interface. The main window shows a 3D model of a beam structure with a distributed load applied. The load is represented by a series of downward-pointing blue arrows along the length of the beam. The beam is supported at one end by a pin support and at the other end by a roller support. The software interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar with various icons, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The 'Load & Definition' dialog box is open on the right side of the screen, showing the following details:

- Definitions:** 1: CV+CM
- Load Cases Details:** SELFWEIGHT Y -1, UNI GY -281 kg/m
- Load Envelopes:** (empty)

At the bottom of the dialog box, there are buttons for 'New...', 'Add...', 'Edit...', and 'Delete...'. Below these buttons, there is a checkbox for 'Toggle Load' and a section for 'Assignment Method' with four radio button options: 'Assign To Selected Beams/Plates', 'Use Cursor To Assign' (which is selected), 'Assign To View', and 'Assign To Edit List'. A text input field contains the number '1'. At the bottom of the dialog box, there are buttons for 'Assigning', 'Close', and 'Help'.

At the bottom of the software window, the status bar displays 'Modeling Mo: Load 1: CV+CM' and 'Input Units: kg-m'. A small coordinate system icon is visible in the bottom left corner of the main window.

ANALYSIS/PRINT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-5 software interface. The main window shows a 2D plot of a structure with two nodes connected by a member, labeled "Load 1". The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The left sidebar contains icons for Setup, Geometry, Analysis, General, Post-Print, Analysis/Print, and Design. The right sidebar shows the "Analysis - Whole Structure" tree view, which includes the following items:

- START JOB INFORMATION
- INPUT WIDTH 79
- UNIT METER KG
- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CV+CM
 - SELFWEIGHT Y -1
 - MEMBER LOAD
 - UNI GY -281
- PERFORM ANALYSIS
- FINISH

At the bottom of the interface, the status bar displays "For Help, press F1", "Modeling Mo Load 1 : CV+CM", and "Input Units: kg-m".

DESIGN

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - TRABE T-5 software interface. The main window shows a 2D model of a structure with a diagonal member and a support. The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help), a toolbar, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The Concrete Design tab is active, showing a tree view of design steps and a list of design parameters.

Concrete Design - Whole Structure

Current Code: Mexican

- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CV+CM
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Highlight Assigned Geometry
 Toggle Assign

Select Parameters... Define Parameters... Commands...

Assignment Method

- Assign To Selected Beams
- Assign To View
- Use Cursor To Assign
- Assign To Edit List

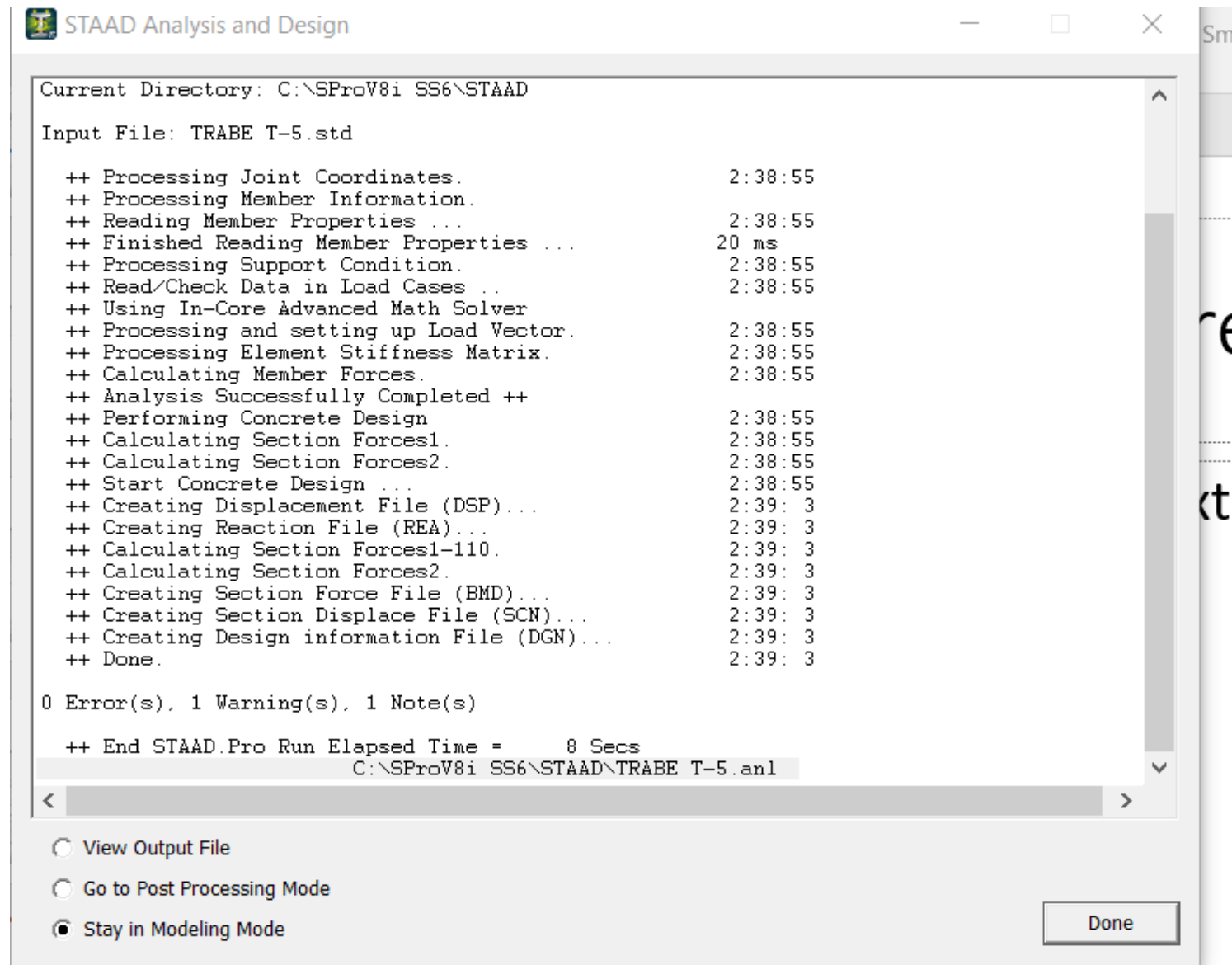
Select Group/Deck

Assign Close Help

For Help, press F1

Modeling Mo Load 1 : CV+CM Input Units: kg-m

ANALISE



STAAD Analysis and Design

Current Directory: C:\SProV8i SS6\STAAD

Input File: TRABE T-5.std

```
++ Processing Joint Coordinates.                2:38:55
++ Processing Member Information.
++ Reading Member Properties ...                2:38:55
++ Finished Reading Member Properties ...      20 ms
++ Processing Support Condition.                2:38:55
++ Read/Check Data in Load Cases ...          2:38:55
++ Using In-Core Advanced Math Solver
++ Processing and setting up Load Vector.      2:38:55
++ Processing Element Stiffness Matrix.        2:38:55
++ Calculating Member Forces.                  2:38:55
++ Analysis Successfully Completed ++
++ Performing Concrete Design                  2:38:55
++ Calculating Section Forces1.                2:38:55
++ Calculating Section Forces2.                2:38:55
++ Start Concrete Design ...                   2:38:55
++ Creating Displacement File (DSP)...          2:39: 3
++ Creating Reaction File (REA)...             2:39: 3
++ Calculating Section Forces1-110.            2:39: 3
++ Calculating Section Forces2.                2:39: 3
++ Creating Section Force File (BMD)...        2:39: 3
++ Creating Section Displace File (SCN)...     2:39: 3
++ Creating Design information File (DGN)...    2:39: 3
++ Done.                                       2:39: 3
```

0 Error(s), 1 Warning(s), 1 Note(s)

++ End STAAD.Pro Run Elapsed Time = 8 Secs
C:\SProV8i SS6\STAAD\TRABE T-5.anl

View Output File
 Go to Post Processing Mode
 Stay in Modeling Mode

Done

ANALISE

TRABE T-5.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

BEAM NO. 1 DESIGN RESULTS - FLEXURE

PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES,DDF

LEN - 2150.00 (mm) FY - 412. FC - 20. SIZE - 150.00 X 200.00 (mm)

| LEVEL | HEIGHT (mm) | BAR INFO | FROM (mm) | TO (mm) | ANCHOR | |
|-------|----------------|----------|--------------|------------|--------|-----|
| | | | | | STA | END |
| 1 | 42. | 2 - 2.MM | 0. | 2113. | YES | NO |

-----|
| CRITICAL POS MOMENT= 2.00 kNm AT 1075.00 (mm) LOAD 1 |
| REQD STEEL= 46.46 (mm2) ROW=0.0020 ROWMX=0.0152 ROWMN=0.0018 |
| REQD COMP STEEL= 0.00 (mm2) |
| MAX/MIN/ACTUAL BAR SPACING= 66.22/ 37.90/ 66.22 (mm) |
| COMP MAX/MIN/ACTUAL BAR SPACING= 0.00/ 0.00/ 0.00 (mm) |
| BASIC/REQD. DEVELOPMENT LENGTH = 199.08/ 178.30 (mm) |
-----|

Cracked Moment of Inertia I_z at above location =0.13657E+08 mm⁴

REQUIRED REINF. STEEL SUMMARY :

-----|
SECTION REINF STEEL (+VE/-VE) MOMENTS (+VE/-VE) LOAD (+VE/-VE)
-----|

Notes: NOTES RESULTS

Total Page: 4 CAP NUM

ANALISE

TRABE T-5.anl - STAAD Output Viewer

File Edit View Help

WARNING

***WARNING - INSTABILITY AT JOINT

Cracked Moment of Inertia I_z at above location =0.13657E+08 mm⁴

REQUIRED REINF. STEEL SUMMARY :

| SECTION (MM) | REINF STEEL (+VE/-VE) (SQ. MM) | MOMENTS (+VE/-VE) (KNS-MET) | LOAD (+VE/-VE) |
|-------------------|------------------------------------|---------------------------------|----------------|
| 0.00 | 0.00/ 0.00 | 0./ 0.00 | 1/ 0 |
| 179.17 | 39.29/ 0.00 | 1./ 0.00 | 1/ 0 |
| 358.33 | 39.29/ 0.00 | 1./ 0.00 | 1/ 0 |
| 537.50 | 39.29/ 0.00 | 2./ 0.00 | 1/ 0 |
| 716.67 | 45.75/ 0.00 | 2./ 0.00 | 1/ 0 |
| 895.83 | 50.16/ 0.00 | 2./ 0.00 | 1/ 0 |
| 1075.00 | 50.56/ 0.00 | 2./ 0.00 | 1/ 0 |
| 1254.17 | 50.56/ 0.00 | 2./ 0.00 | 1/ 0 |
| 1433.33 | 50.56/ 0.00 | 2./ 0.00 | 1/ 0 |
| 1612.50 | 50.56/ 0.00 | 2./ 0.00 | 1/ 0 |
| 1791.67 | 50.56/ 0.00 | 1./ 0.00 | 1/ 0 |
| 1970.83 | 50.56/ 0.00 | 1./ 0.00 | 1/ 0 |
| 2150.00 | 0.00/ 0.00 | 0./ 0.00 | 0/ 1 |

BEAM NO. 1 DESIGN RESULTS - SHEAR

NOTES

RESULTS

Total Page : 4 CAP NUM

ANALISE

TRABE T-5.anl - STAAD Output Viewer

File Edit View Help

***WARNING - INSTABILITY AT JOINT

```
BEAM NO. 1 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu= 0.01 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1
STIRRUPS ARE NOT REQUIRED.

-----< PAGE 3 Ends Here >-----

STAAD SPACE -- PAGE NO. 4

AT END SUPPORT - Vu= 0.01 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1
STIRRUPS ARE NOT REQUIRED.

37. END CONCRETE DESIGN
38. FINISH

***** END OF THE STAAD.Pro RUN *****

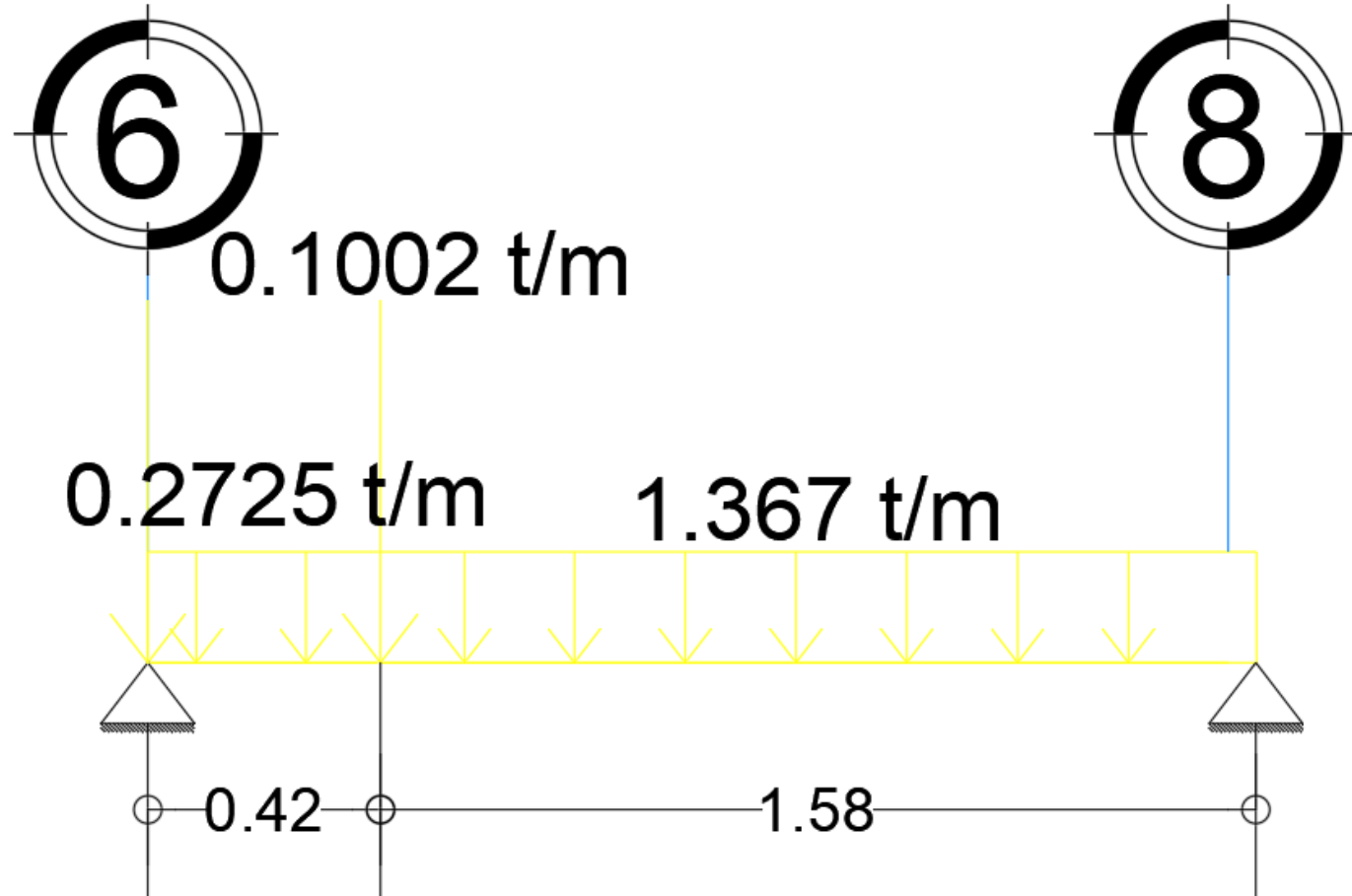
**** DATE= FEB 17,2021 TIME= 2:39: 3 ****

*****
* For technical assistance on STAAD.Pro, please visit *
* http://selectservices.bentley.com/en-US/ *
* *
* Details about additional assistance from *
* Bentley and Partners can be found at program menu *
* Help->Technical Support *
```

NOTES

RESULTS

Total Page: 4 CAP NUM



TRABE T11

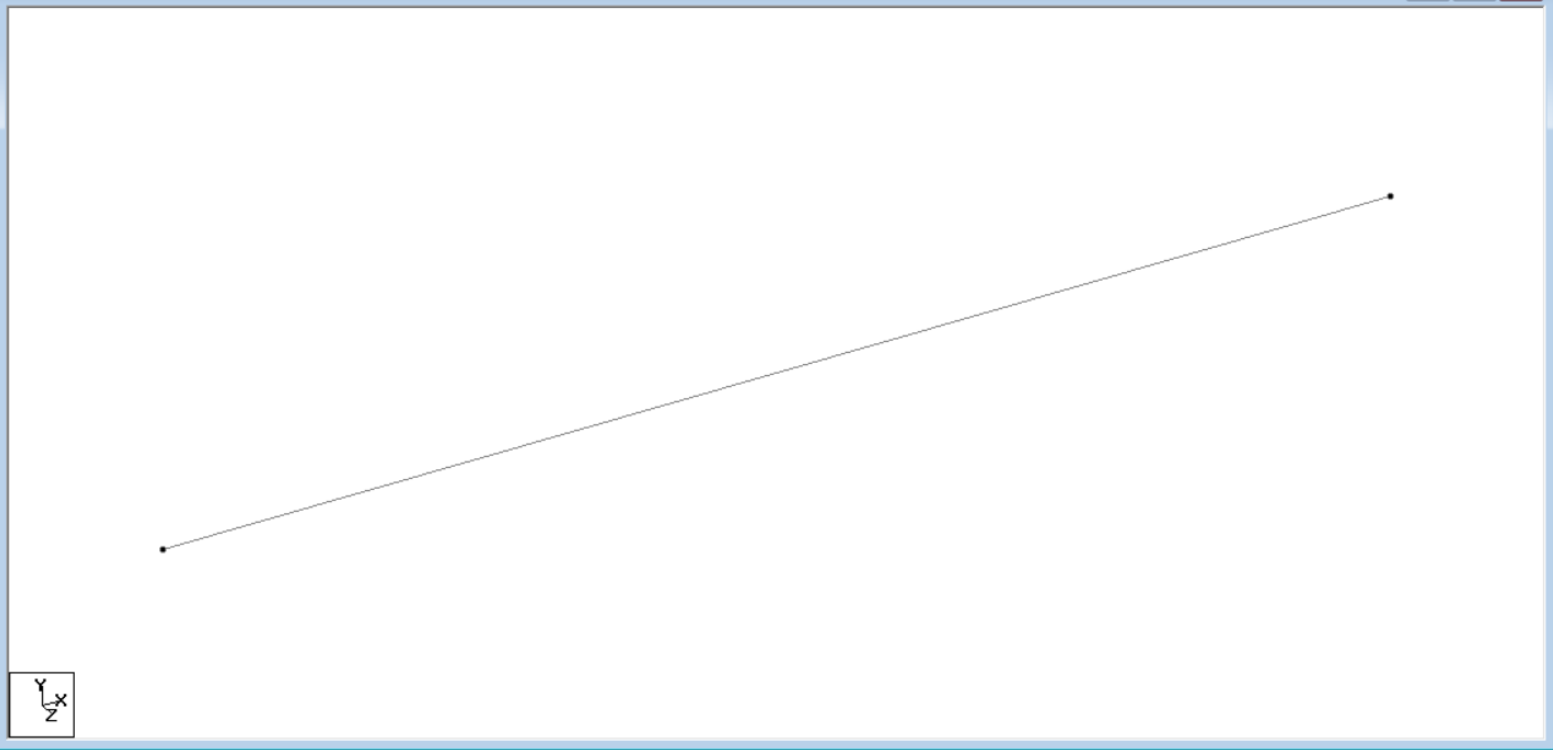
GEOMETRY

STAAD.Pro V8i (SELECTseries 6) - trabe t-11

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

trabe t-11 - Whole Structure



trabe t-11 - Nodes

| Node | X m | Y m | Z m |
|------|--------|--------|--------|
| 1 | 0.000 | 3.000 | 0.000 |
| 2 | 2.000 | 3.000 | 0.000 |
| 3 | | | |

trabe t-11 - Beams

| Beam | Node A | Node B | Property Refn. |
|------|--------|--------|----------------|
| 1 | 1 | 2 | |
| 2 | | | |

Click on node at start of beam

Modeling Mo

Input Units: kg-m

GENERAL

STAAD.Pro V8i (SELECTseries 6) - trabe t-11

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

trabe t-11 - Whole Structure

trabe t-11 - Beams

| Beam | Node A | Node B | Property Refn. | Mate |
|------|--------|--------|----------------|-------|
| 1 | 1 | 2 | 1 | CONCR |
| 2 | | | | |

Properties - Whole Structure

Section Beta Angle

| Ref | Section | Material |
|-----|----------------|----------|
| 1 | Rect 0.20x0.15 | CONCRETE |

Highlight Assigned Geometry

Edit... Delete...

Values... Section Database Define...

Materials... Thickness... User Table...

Assignment Method

Assign To Selected Beams Use Cursor To Assign

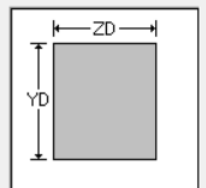
Assign To Edit List Assign To View

1

Assign Close Help

Property

Rectangle



YD: 0.2 m

ZD: 0.15 m

Material

CONCRETE

Change Assign Close Help

For Help, press F1

Modeling Mo

Input Units: kq-m

SUPPORT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) software interface. The main window shows a 2D model of a structure with two support points, represented by blue pin icons, connected by a diagonal line. The interface includes a menu bar (File, Edit, View, Tools, Select, Geometry, Commands, Analyze, Mode, Bentley Cloud Services, Window, Help) and a toolbar with various icons for modeling and analysis. The 'Supports' dialog box is open, showing a table of node supports and an 'Assignment Method' section.

trabe t-11 - Node Supports

| Node | Support | Description |
|------|---------|-------------|
| 1 | S2 | Support 2 |
| 2 | S2 | Support 2 |

Supports - Whole Structure

| Ref | Description |
|-----|-------------|
| S1 | No support |
| S2 | Support 2 |

Assignment Method

Assign To Selected Nodes

Assign To View

Use Cursor To Assign

Assign To Edit List

12

Assign Close Help

For Help, press F1

Modeling Mo

Input Units: kg-m

LOAD Y DEFINITION

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - trabe t-11 interface. The main window shows a 3D model of a structure with a load definition window open on the right. The load definition window is titled "Load & Definition" and contains the following information:

- Definitions**
 - Load Cases Details**
 - 1 : CM+CV
 - SELFWEIGHT Y -1
 - UNI GY -1367.7 0.42 kg/m
 - UNI GY -272.542 0 0.42 kg/m
 - CON GY -100.24 0.42 kg/m
 - Load Envelopes**

The load definition window also includes buttons for "New...", "Add...", "Edit...", and "Delete...". Below these buttons, there is a "Toggle Load" checkbox and an "Assignment Method" section with radio buttons for "Assign To Selected Beams/Plates", "Use Cursor To Assign" (selected), "Assign To View", and "Assign To Edit List". A text box contains the value "1". At the bottom of the window are "Assign", "Close", and "Help" buttons.

The main window shows a 3D model of a structure with a load definition window open. The load is defined as "Load 1" and is applied to the structure. The load definition window is titled "Load & Definition" and contains the following information:

- Definitions**
 - Load Cases Details**
 - 1 : CM+CV
 - SELFWEIGHT Y -1
 - UNI GY -1367.7 0.42 kg/m
 - UNI GY -272.542 0 0.42 kg/m
 - CON GY -100.24 0.42 kg/m
 - Load Envelopes**

The load definition window also includes buttons for "New...", "Add...", "Edit...", and "Delete...". Below these buttons, there is a "Toggle Load" checkbox and an "Assignment Method" section with radio buttons for "Assign To Selected Beams/Plates", "Use Cursor To Assign" (selected), "Assign To View", and "Assign To Edit List". A text box contains the value "1". At the bottom of the window are "Assign", "Close", and "Help" buttons.

ANALYSIS/PRINT

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) software interface. The main window shows a 2D model of a structure with a single member and two supports. The interface includes a menu bar, a toolbar, and a ribbon with tabs for Modeling, Building Planner, Piping, Bridge Deck, Postprocessing, Foundation Design, Steel Design, RAM Connection, Concrete Design, Advanced Slab Design, and Earthquake. The Analysis/Print ribbon is active, showing options for Setup, Geometry, Analysis, Pre-Print, General, Post-Print, and Design. The Analysis/Print window is open, displaying a tree view of the analysis process. The tree view includes the following items:

- START JOB INFORMATION
 - INPUT WIDTH 79
 - UNIT METER KG
 - JOINT COORDINATES
 - MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CM+CV
 - SELFWEIGHT Y -1
 - MEMBER LOAD
 - UNI GY -1367.7 0.42
 - UNI GY -272.542 0 0.42
 - CON GY -100.24 0.42
- PERFORM ANALYSIS
- FINISH

The status bar at the bottom indicates "Modeling Mo Load 1 : CM+CV" and "Input Units: kg-m".

DESIGN

STAAD.Pro V8i (SELECTSeries 6) - trabe t-11

File Edit View Tools Select Geometry Commands Analyze Mode Bentley Cloud Services Window Help

Modeling Building Planner Piping Bridge Deck Postprocessing Foundation Design Steel Design RAM Connection Concrete Design Advanced Slab Design Earthquake

trabe t-11 - Whole Structure

Concrete Design - Whole Structure

Current Code: Mexican

- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- CONSTANTS
- SUPPORTS
- LOAD 1 LOADTYPE None TITLE CM+CV
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

Highlight Assigned Geometry

Toggle Assign

Select Parameters... Define Parameters... Commands...

Assignment Method

Assign To Selected Beams

Assign To View

Use Cursor To Assign

Assign To Edit List

Select Group/Deck

1

Assign Close Help

Load 1

For Help, press F1

Modeling Mo: Load 1: CM+CV Input Units: kg-m

ANALISE

The screenshot displays the STAAD.Pro V8i (SELECTseries 6) - trabe t-11 interface. The main window shows a 3D model of a structure with a load applied. A 'STAAD Analysis and Design' dialog box is open, displaying the following log:

```
Current Directory: C:\SProV8i SS6\STAAD
Input File: trabe t-11.std
++ Processing Joint Coordinates.          16: 3:11
++ Processing Member Information.        16: 3:11
++ Reading Member Properties ...         16: 3:11
++ Finished Reading Member Properties ... 10 ms
++ Processing Support Condition.         16: 3:11
++ Read/Check Data in Load Cases ...    16: 3:11
++ Using In-Core Advanced Math Solver
++ Processing and setting up Load Vector. 16: 3:11
++ Processing Element Stiffness Matrix.  16: 3:11
++ Calculating Member Forces            16: 3:11
++ Analysis Successfully Completed ++
++ Performing Concrete Design            16: 3:11
++ Calculating Section Forces1.         16: 3:11
++ Calculating Section Forces2.         16: 3:11
++ Start Concrete Design ...            16: 3:11
++ Creating Displacement File (DSP)....  16: 3:15
++ Creating Reaction File (REA)....      16: 3:15
++ Calculating Section Forces1-110.     16: 3:15
++ Calculating Section Forces2.         16: 3:15
++ Creating Section Force File (BMD).... 16: 3:15
++ Creating Section Displace File (SCN)... 16: 3:15
++ Creating Design information File (DGN)... 16: 3:15
++ Done.                                 16: 3:15

0 Error(s), 2 Warning(s), 1 Note(s)
++ End STAAD.Pro Run Elapsed Time =      4 Secs
C:\SProV8i SS6\STAAD\trabe t-11.enl
```

Below the log, the 'Stay in Modeling Mode' option is selected. The 'Concrete Design - Whole Structure' dialog box is also open, showing the 'Current Code: Mexican' and a tree view of design options:

- JOINT COORDINATES
- MEMBER INCIDENCES
- DEFINE MATERIAL START
- MEMBER PROPERTY
- SUPPORTS
- CONSTANTS
- LOAD 1 LOADTYPE None TITLE CM+CV
- PERFORM ANALYSIS
- START CONCRETE DESIGN
 - CODE MEXICAN
 - FC 2.00014e+006
 - TRACK 2
 - DESIGN BEAM
 - END CONCRETE DESIGN
- FINISH

The 'Highlight Assigned Geometry' checkbox is checked. The 'Assignment Method' section has 'Use Cursor To Assign' selected. The 'Assign' button is visible at the bottom.

ANALISE

trabe t-11.anl - STAAD Output Viewer

File Edit View Help

WARNING

WARNING: IN UNIFORM MEMBE
***WARNING - INSTABILITY AT J

NOTES

RESULTS

BEAM NO. 1 DESIGN RESULTS - FLEXURE

PER CODE NTC FOR THE DESIGN AND CONSTRUCTION OF CONCRETE STRUCTURES,DDF

LEN - 2000.00 (mm) FY - 412. FC - 20. SIZE - 150.00 X 200.00 (mm)

| LEVEL | HEIGHT (mm) | BAR INFO | FROM (mm) | TO (mm) | ANCHOR STA END |
|-------|----------------|----------|--------------|------------|-------------------|
| 1 | 43. | 2 - 3MM | 0. | 2000. | YES YES |

-----|
| CRITICAL POS MOMENT= 6.79 kNm AT 1000.00 (mm) LOAD 1|
| REQD STEEL= 127.04 (mm2) ROW=0.0054 ROWMX=0.0152 ROWMN=0.0018 |
| REQD COMP STEEL= 0.00 (mm2) |
| MAX/MIN/ACTUAL BAR SPACING= 64.63/ 39.50/ 64.63 (mm) |
| COMP MAX/MIN/ACTUAL BAR SPACING= 0.00/ 0.00/ 0.00 (mm) |
| BASIC/REQD. DEVELOPMENT LENGTH = 239.40/ 268.40 (mm) |
|-----|

Cracked Moment of Inertia I_z at above location =0.18320E+08 mm⁴

REQUIRED REINF. STEEL SUMMARY :

| SECTION (MM) | REINF STEEL(+VE/-VE) (SQ. MM) | MOMENTS (+VE/-VE) (KNS-MET) | LOAD (+VE/-VE) |
|-------------------|-----------------------------------|---------------------------------|----------------|
|-------------------|-----------------------------------|---------------------------------|----------------|

Total Page: 4 CAP NUM

ANALISE

trabe t-11.anl - STAAD Output Viewer

File Edit View Help

WARNING

WARNING: IN UNIFORM MEMBE
***WARNING - INSTABILITY AT J

NOTES

RESULTS

REQUIRED REINF. STEEL SUMMARY :

| SECTION (MM) | REINF STEEL (+VE/-VE) (SQ. MM) | MOMENTS (+VE/-VE) (KNS-MET) | LOAD (+VE/-VE) |
|-------------------|------------------------------------|---------------------------------|----------------|
| 0.00 | 0.00/ | 0.00 | 0/ 1 |
| 166.67 | 45.34/ | 0.00 | 2/ 0 |
| 333.33 | 68.07/ | 0.00 | 3/ 0 |
| 500.00 | 99.43/ | 0.00 | 5/ 0 |
| 666.67 | 122.44/ | 0.00 | 6/ 0 |
| 833.33 | 137.12/ | 0.00 | 7/ 0 |
| 1000.00 | 142.81/ | 0.00 | 7/ 0 |
| 1166.67 | 139.22/ | 0.00 | 7/ 0 |
| 1333.33 | 126.54/ | 0.00 | 6/ 0 |
| 1500.00 | 105.34/ | 0.00 | 5/ 0 |
| 1666.67 | 76.52/ | 0.00 | 4/ 0 |
| 1833.33 | 50.56/ | 0.00 | 2/ 0 |
| 2000.00 | 0.00/ | 0.00 | 0/ 1 |

BEAM NO. 1 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu= 0.02 KN Vc= 0.00 KN Vs= 0.00 KN
Tu= 0.00 Kn Me Tc= 0.00 Kn Me Ts= 0.00 Kn Me LOAD 1

STIRRUPS ARE NOT REQUIRED.

-----< PAGE 3 Ends Here >-----

STAAD SPACE

-- PAGE NO. 4

Total Page: 4 CAP NUM

ANALISE

file edit view help

WARNING

WARNING: IN UNIFORM MEMBE
***WARNING - INSTABILITY AT J

NOTES

RESULTS

```
BEAM NO.      1 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu=  0.02 KN  Vc=  0.00 KN  Vs=  0.00 KN
Tu=  0.00 Kn Me  Tc=  0.00 Kn Me  Ts=  0.00 Kn Me  LOAD  1
                    STIRRUPS ARE NOT REQUIRED.

-----< PAGE 3 Ends Here >-----

STAAD SPACE                                -- PAGE NO.   4

AT END  SUPPORT - Vu=  0.03 KN  Vc=  0.00 KN  Vs=  0.00 KN
Tu=  0.00 Kn Me  Tc=  0.00 Kn Me  Ts=  0.00 Kn Me  LOAD  1
                    STIRRUPS ARE NOT REQUIRED.

39. END CONCRETE DESIGN
40. FINISH

***** END OF THE STAAD.Pro RUN *****

**** DATE= FEB 17,2021  TIME= 16: 3:15 ****

*****
*   For technical assistance on STAAD.Pro, please visit   *
*   http://selectservices.bentley.com/en-US/              *
*                                                         *
*   Details about additional assistance from              *
*   Bentley and Partners can be found at program menu    *
*   Help->Technical Support                               *
*                                                         *
*   Copyright (c) 1997-2015 Bentley Systems, Inc.        *

Total Page: 4 CAP NUM
```