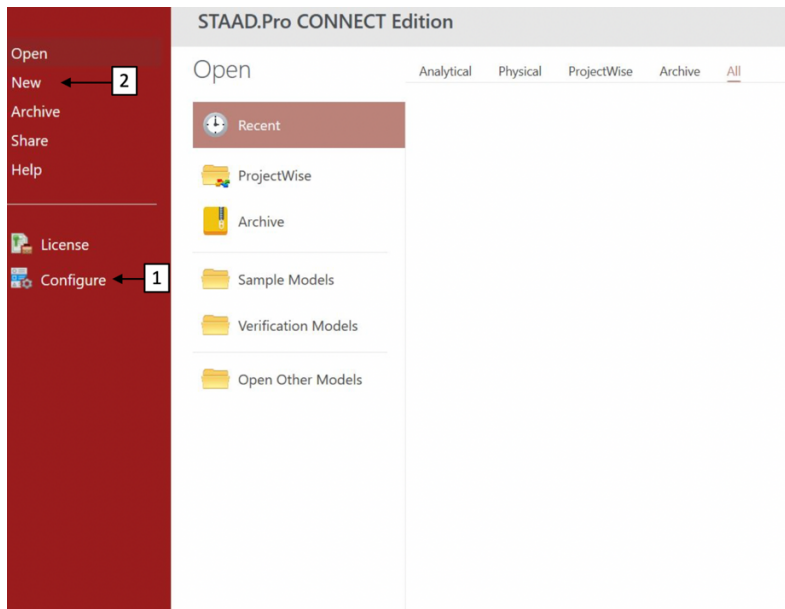
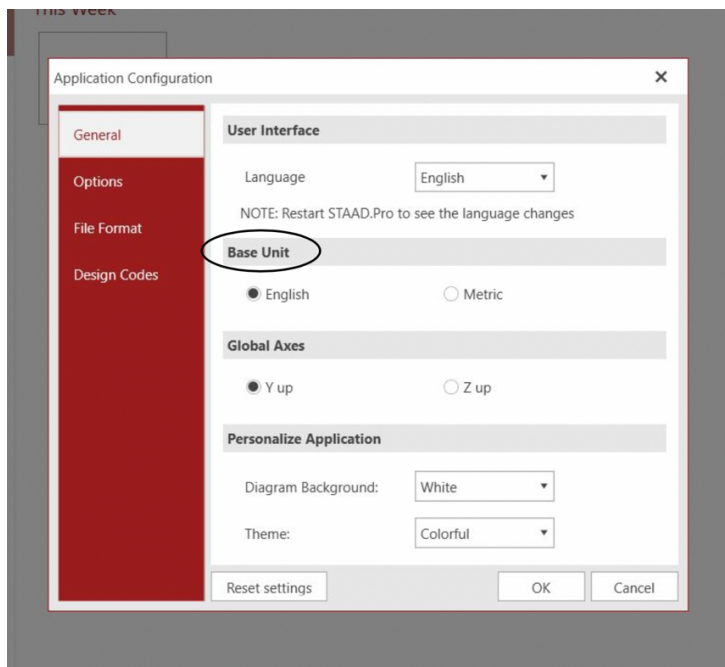


Important Areas of the STAAD Layout

- **Opening Screen**



1. Configure



- Change the unit to match the problem before creating a new project
- The base unit indicates the default input/system unit for the new project

2. New

- Create new project with model and job information

The image displays two screenshots of the STAAD.Pro CONNECT Edition software interface, specifically the 'New' dialog box, illustrating the steps to create a new project.

Top Screenshot: Model Information

- File Name:** Structure1
- Location:** (Empty field with a 'Browse...' button)
- Type:** Analytical (Selected), Physical, Building
- Units:** English (Selected), Metric

Bottom Screenshot: Job Information

- Job Name:** Enter a job name
- Client:** Enter client
- Job Number:** Enter job
- Revision:** Enter revision
- Part:** Enter part
- Reference:** Enter reference
- Engineer:** Name: Enter the engineer's name, Date: Enter date
- Checker:** Name: Enter the checker's name, Date: Enter date
- Approver:** Name: Enter the approver's name, Date: Enter date
- Comments:** Enter comments

CONNECT Information

- Project ID:** Connected Project ID will display here
- Project Name:** Connected Project Name will display here
- Associate Project:** (Button)

1. File Location

- Browse to change the file location to student drive
- Create a folder for each new model is recommended due to STAAD program generates many files

2. Type

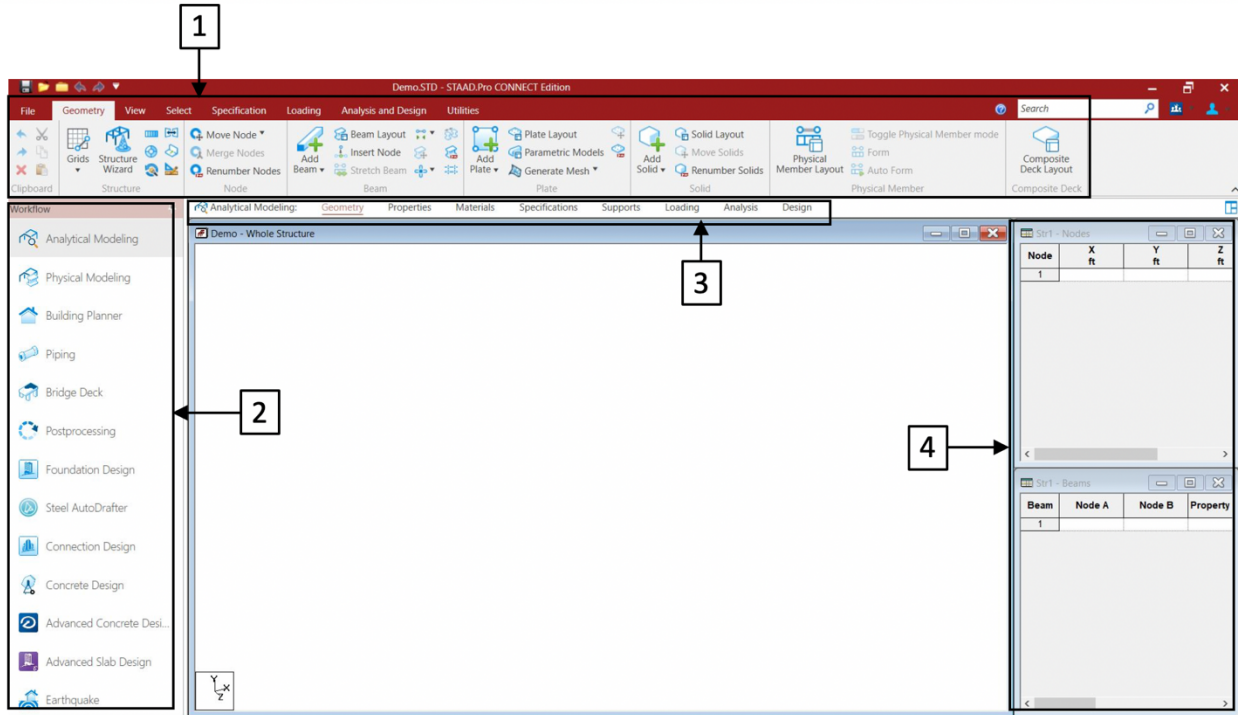
- Use analytical for class exercises, homework and project

- The analytical model is directly processed by the analysis design engine of STAAD program

3. Units

- Check if the unit match with the unit choosing from Configure

- **Graphical User Interface (GUI)**



1. Ribbon Tool Bar

- Provide access to STAAD Pro. modeling, analysis and design commands

2. Workflow Panel

-

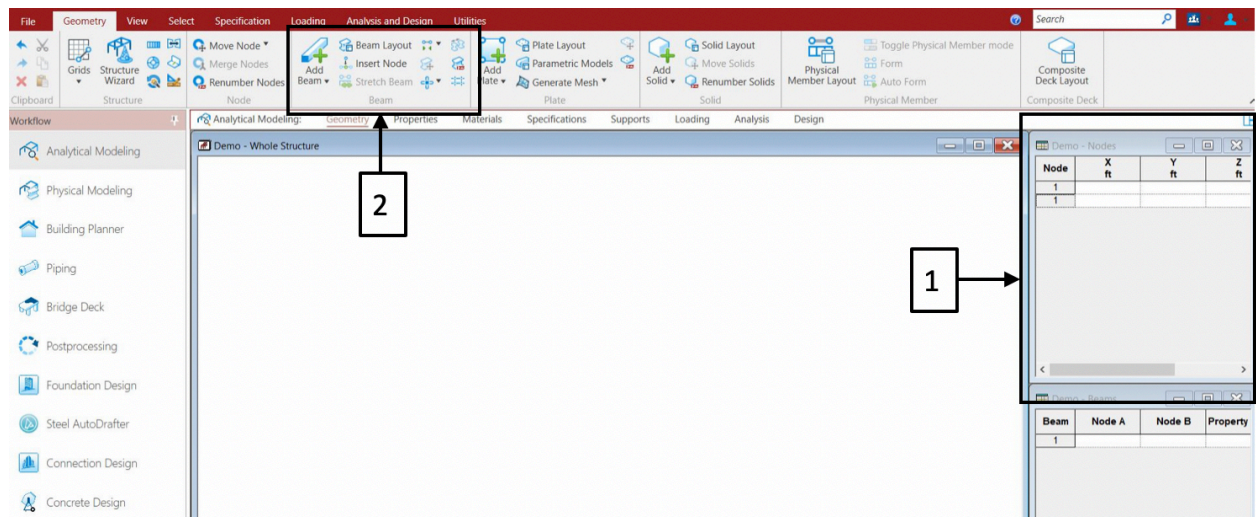
3. Workflow Page Control Area

- The organization of the pages, from left to right, represents the logical sequence of operations (See detailed definition at General Tab section)

4. Data Area

- Contain page dialogs, tables, model status, list boxes, etc.
- appear depending on the type of operation you are performing

- **Geometry Tab**



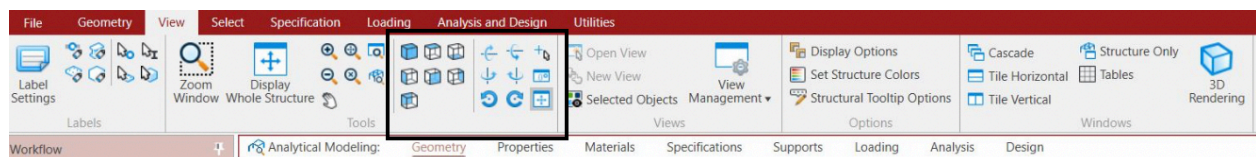
1. Note Window

- Insert node manually by defining the X, Y, Z coordinates

2. Drawing Tools

- Add beams/plates/surfaces/solids – adds an element using existing nodes
- Snap node/beam – Toggles the drawing grid
- Insert node – insert nodes on existing elements
- Generate surface mesh – creates several nodes on a surface for further analysis

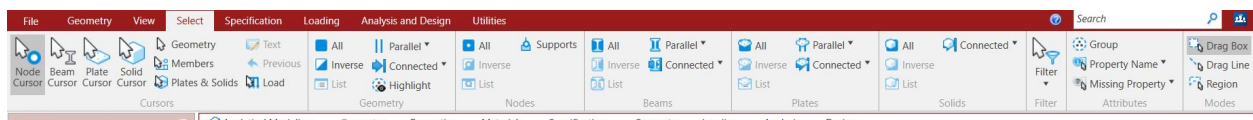
- **View Tab**



- This area allows for manipulation of the model view
- The isometric cubes change the face you view from. The other buttons rotate the model.

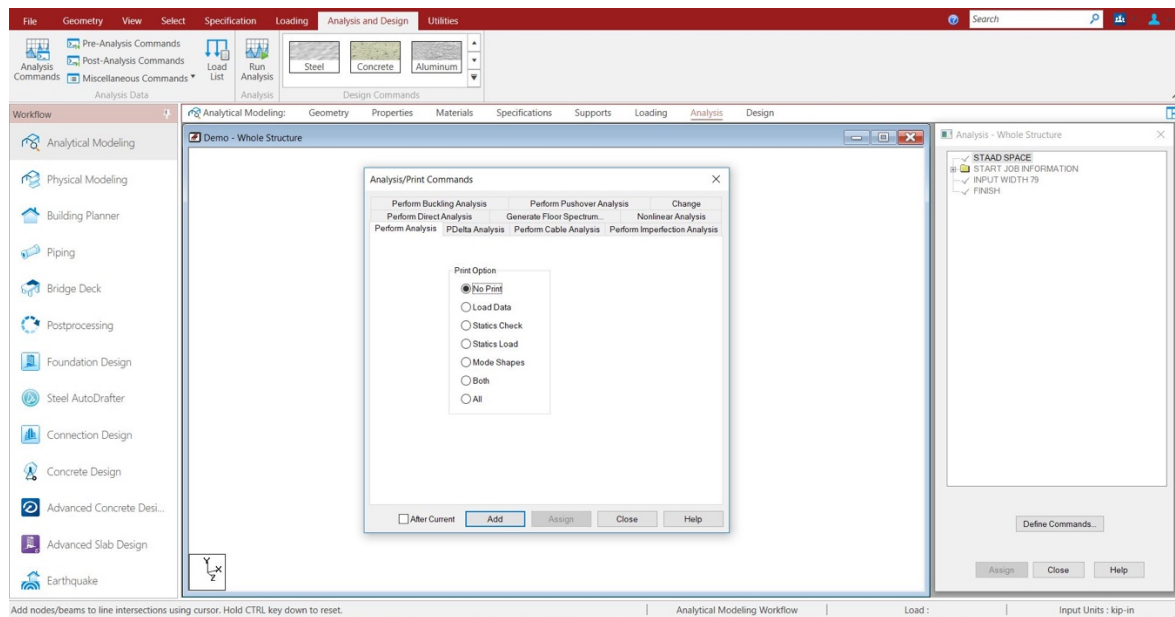
(You can also rotate by right-clicking and dragging)

- **Select Tab**



- Use to change between nodes cursor, beams cursor, plates cursor etc.

- **Analysis & Print Tab**



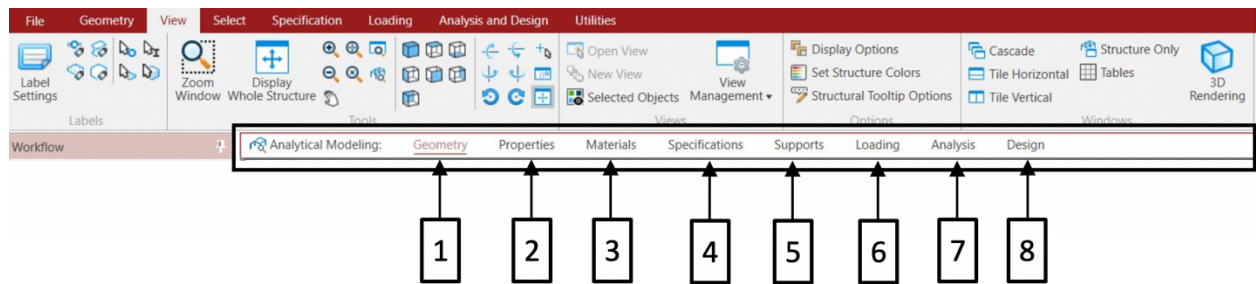
1. **Analysis**

- Add the command “Perform Analysis” with “No Print”

2. **Post-Print**

- Define the commands desired for your output file. These are seen in the tabs above
- After adding the commands, assign them to the members you want the results for

- **General Tab**



1. *Geometry*

- As shown above

2. *Property*

- Define properties for the elements you create (cross-sections, W shapes, plate thicknesses, etc.), also create beta angles (element rotation)

3. *Material*

- Define new materials if structure uses something other than default materials (steel, concrete, aluminum, stainless steel)

4. *Spec*

- Define member releases, offsets, truss members

5. *Support*

- Define supports for the structure: pinned, fixed, rollers, linear/nonlinear springs

6. *Loading*

- Create load cases to define nodal loads, member loads, area loads
- Load generations can be created for moving loads after defining a vehicle

7. *Analysis*

- Add the command “Perform Analysis” with “No Print”

8. *Design*

- Not suing for this class