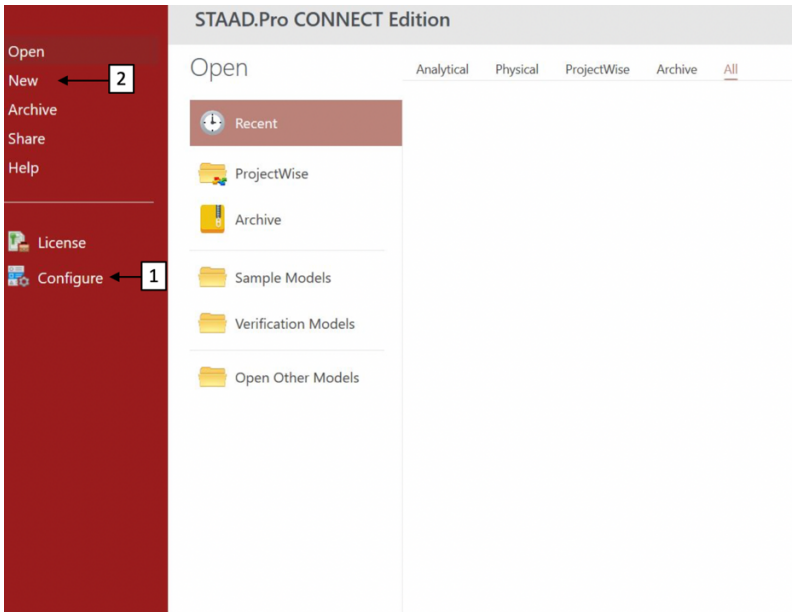
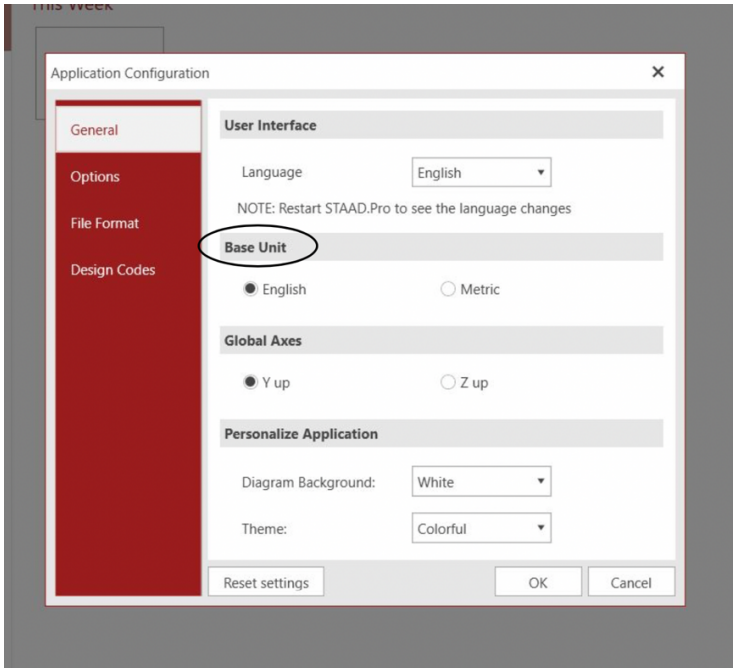
Important Areas of the STAAD Layout

- **Opening Screen**



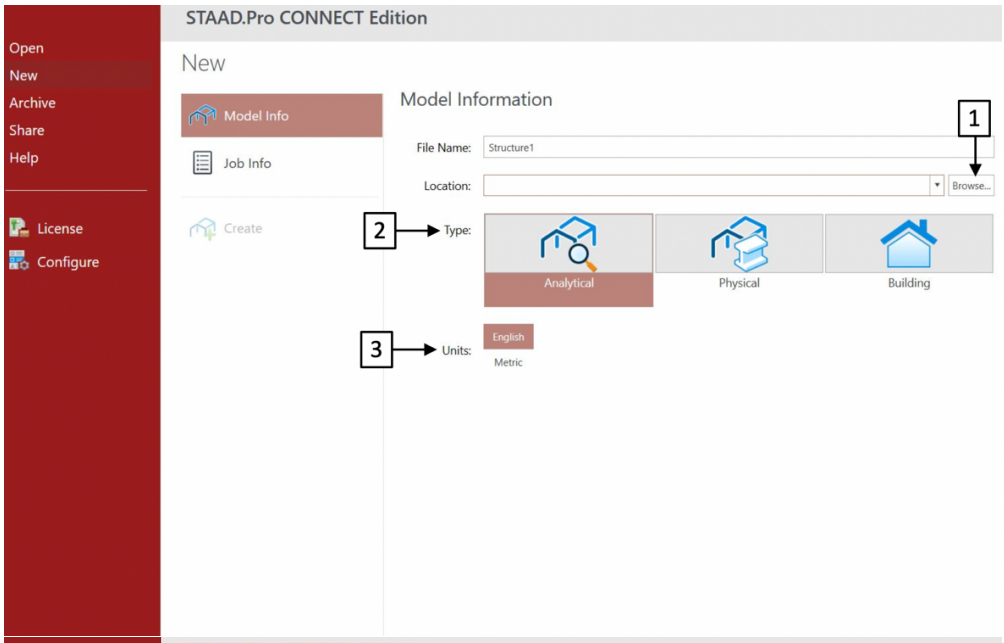
1. *Configure*

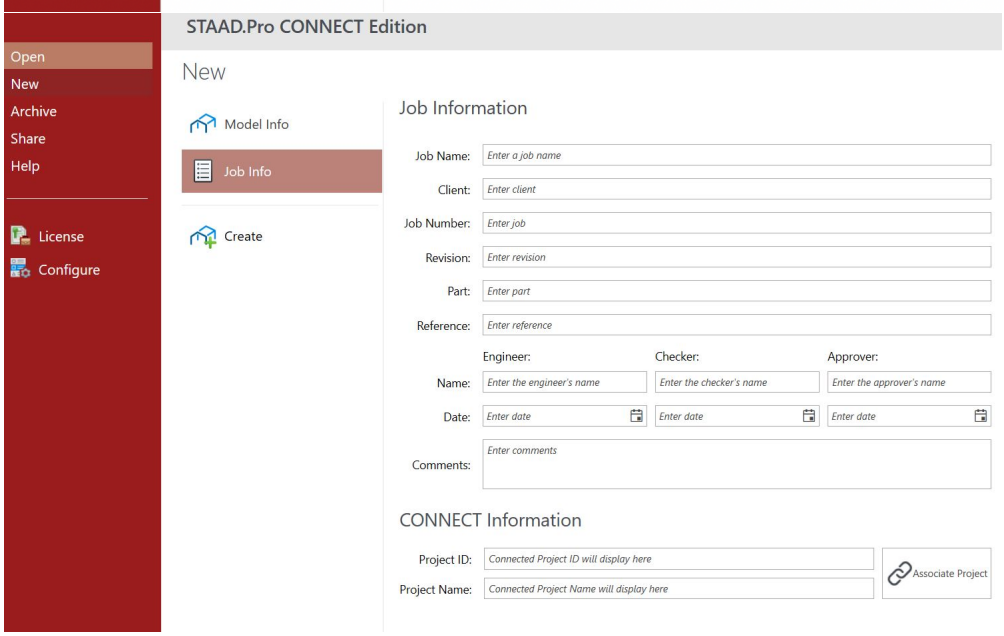


- Change the unit to match the problem before creating a new project
- The base unit indicates the default input/system unit for the new project

2. *New*

- Create new project with model and job information





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*

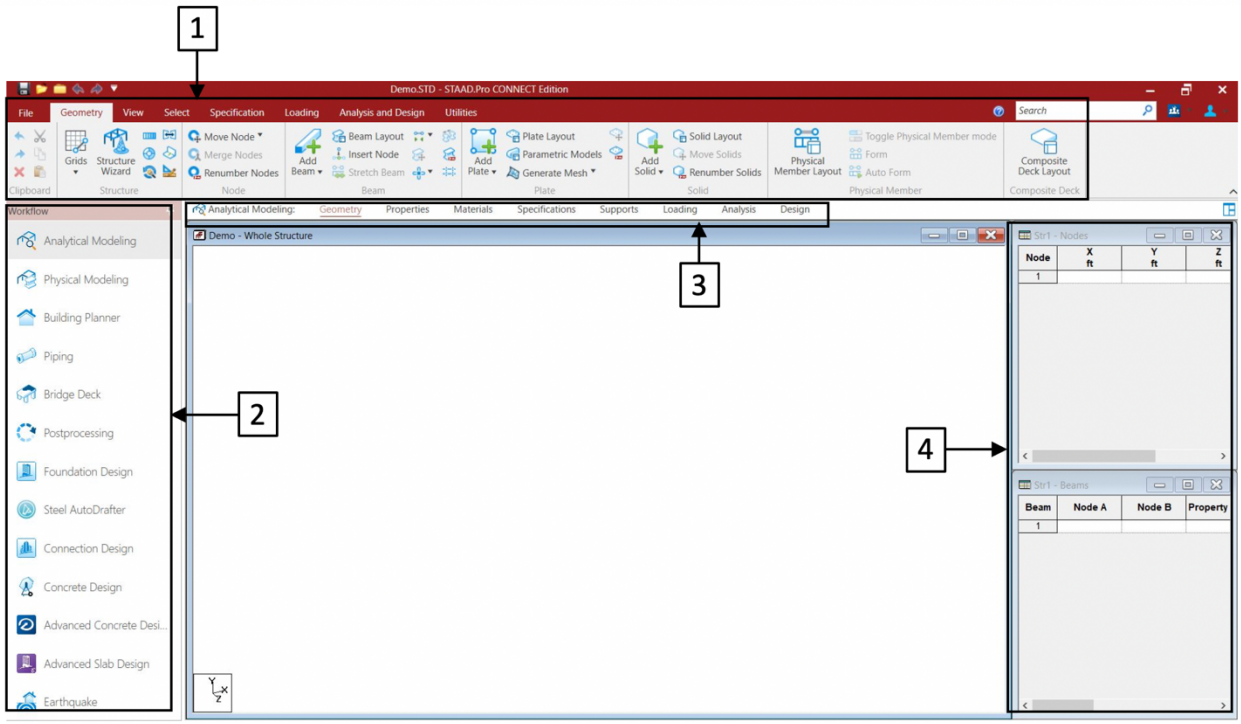
- Use analytical for class exercises, homework and project

- 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*

   -

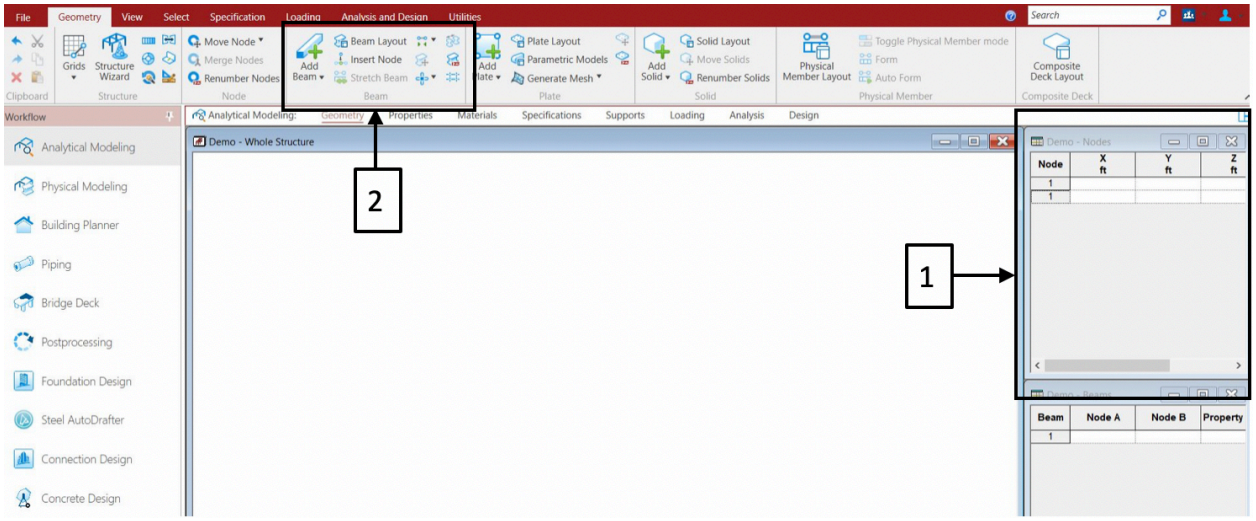
3. *Workflow Page Control Area*

   - 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**



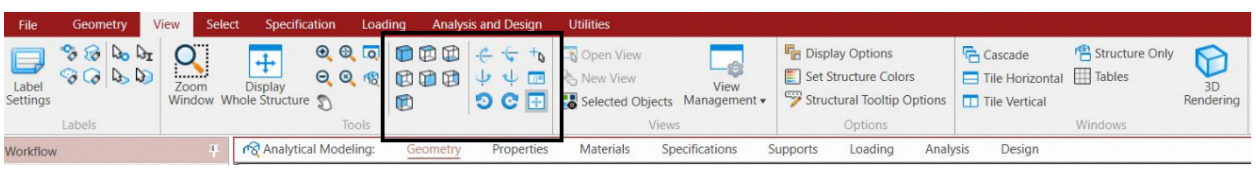
1. *Note Window*

   - 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

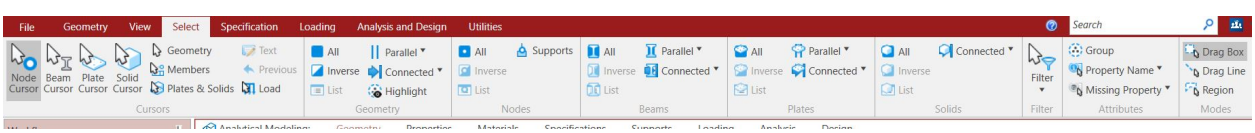
- **View Tab**



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

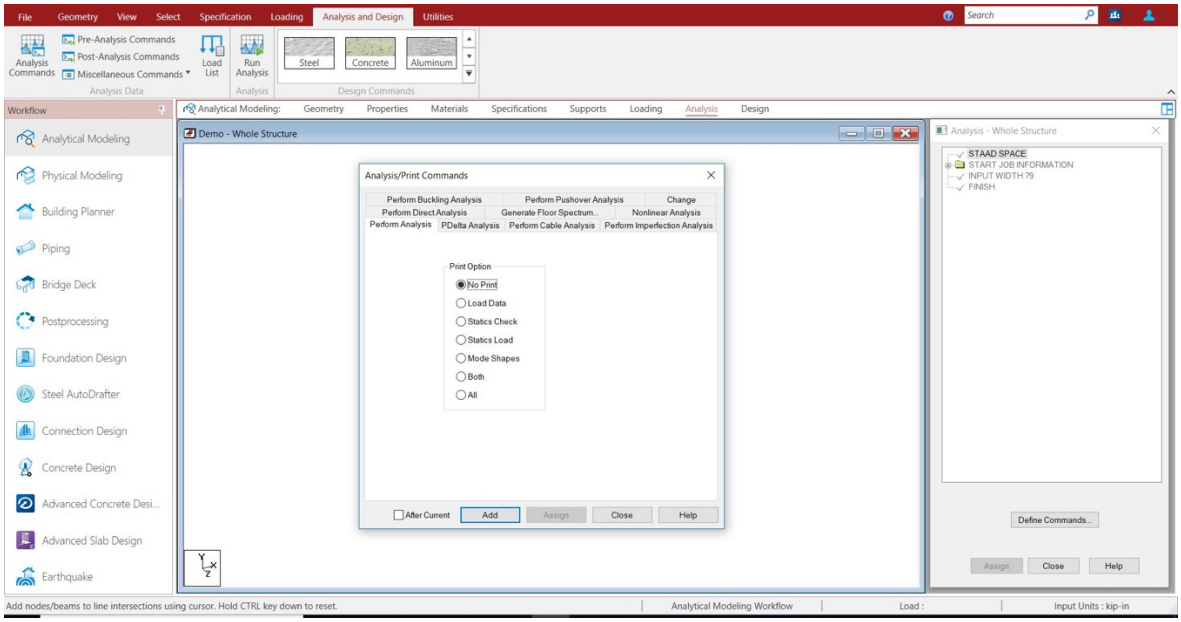
(You can also rotate by right-clicking and dragging)

- **Select Tab**



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

- **Analysis & Print Tab**



1. Analysis

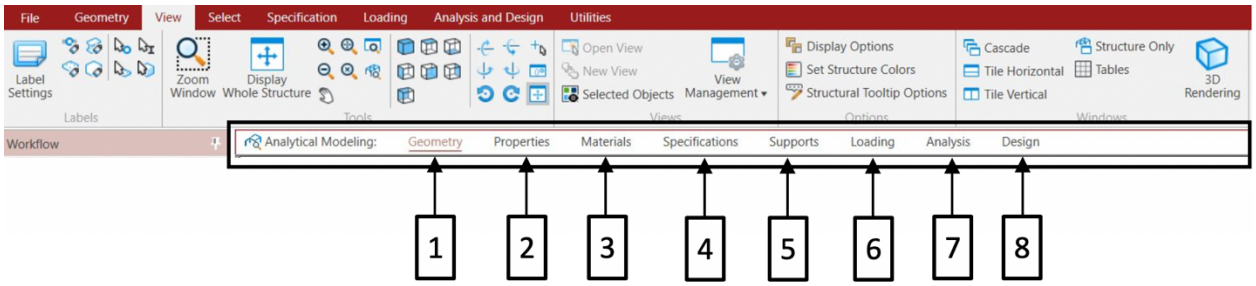
   – Add the command “Perform Analysis” with “No Print”

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