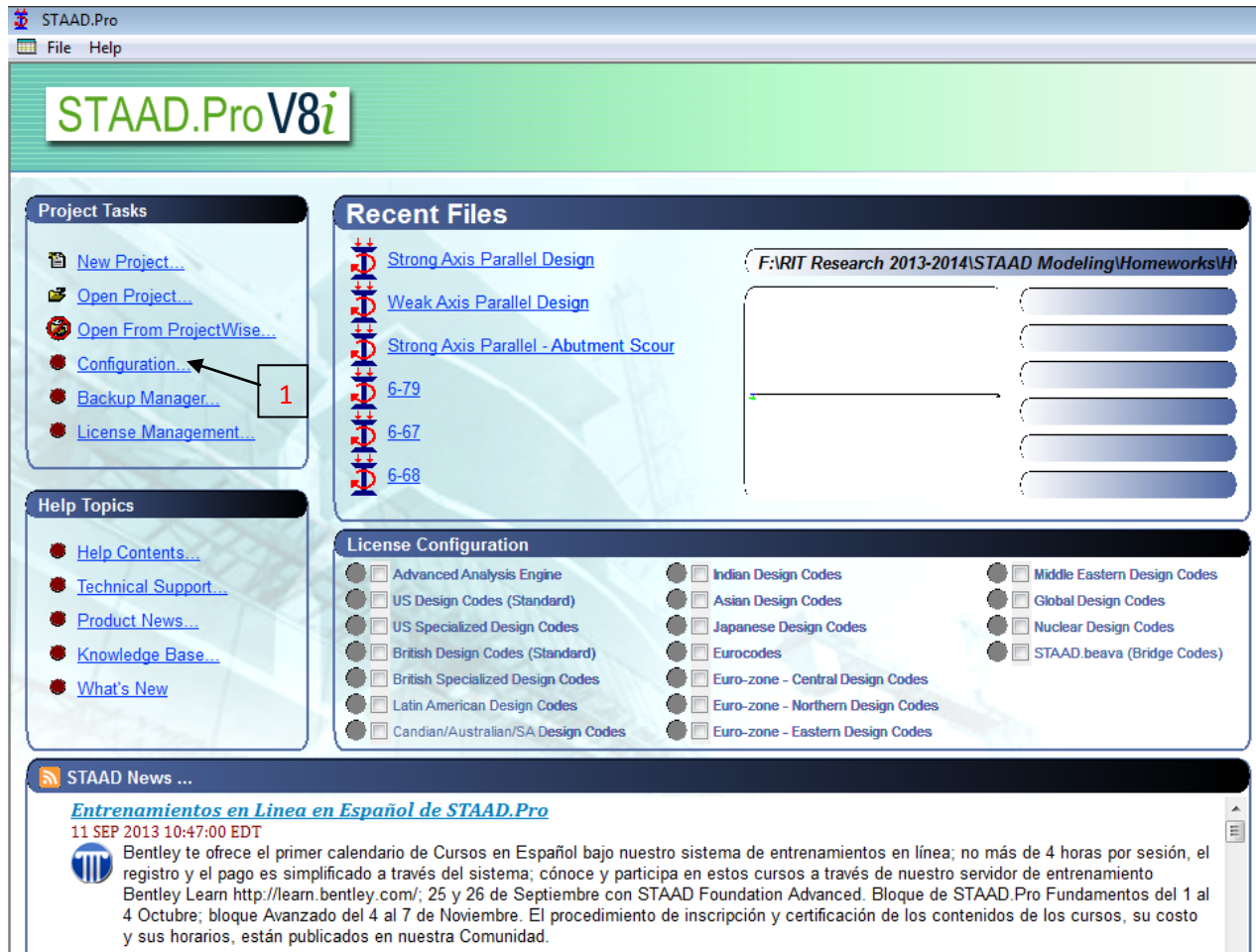


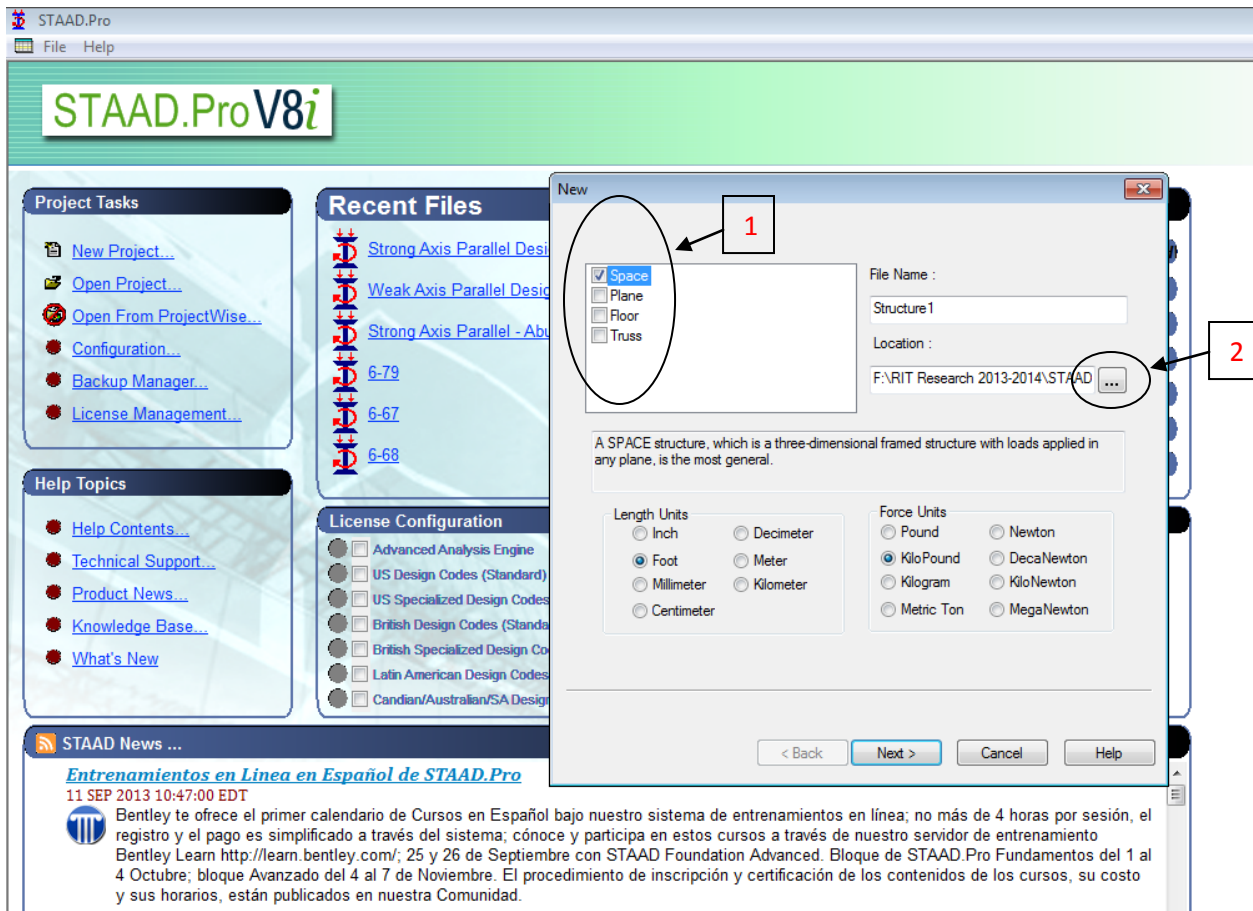
Important areas of the STAAD Layout

- The Opening Screen



1. Configuration – Click here to change the units before opening a new project

- New Project Screen

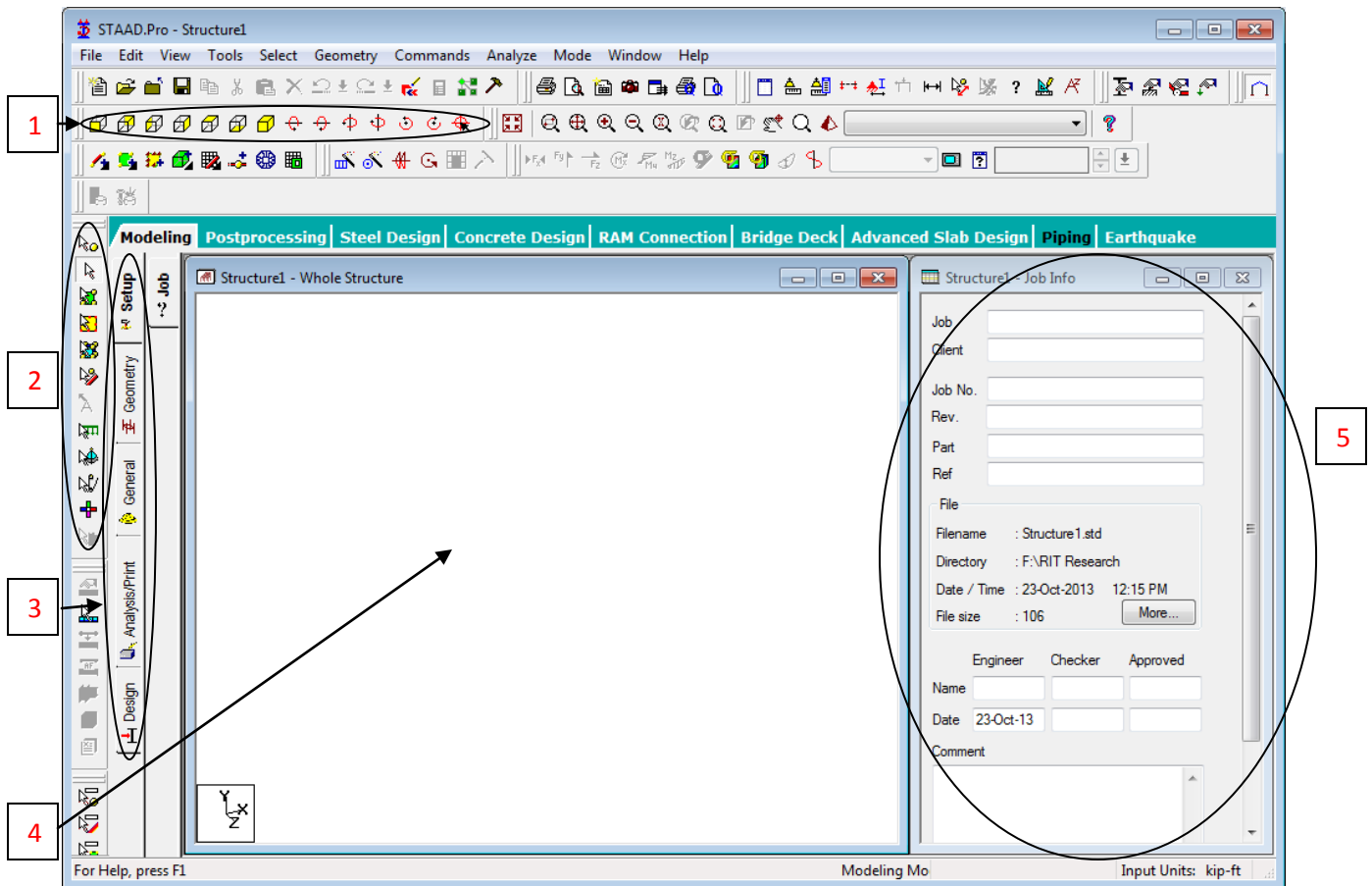


1. Structure Type

- a. Space – Used for 3D modeling
- b. Plane – Used for 2D modeling, geometry, loading and deformation are restricted to the global X-Y plane
- c. Floor - Similar to Plane, but geometry is confined to the X-Z plane
- d. Truss – Used for creating trusses where ALL members are defined as truss members. A Truss structure carries loading by pure axial action. Truss members are deemed incapable of carrying shear, bending and torsion.

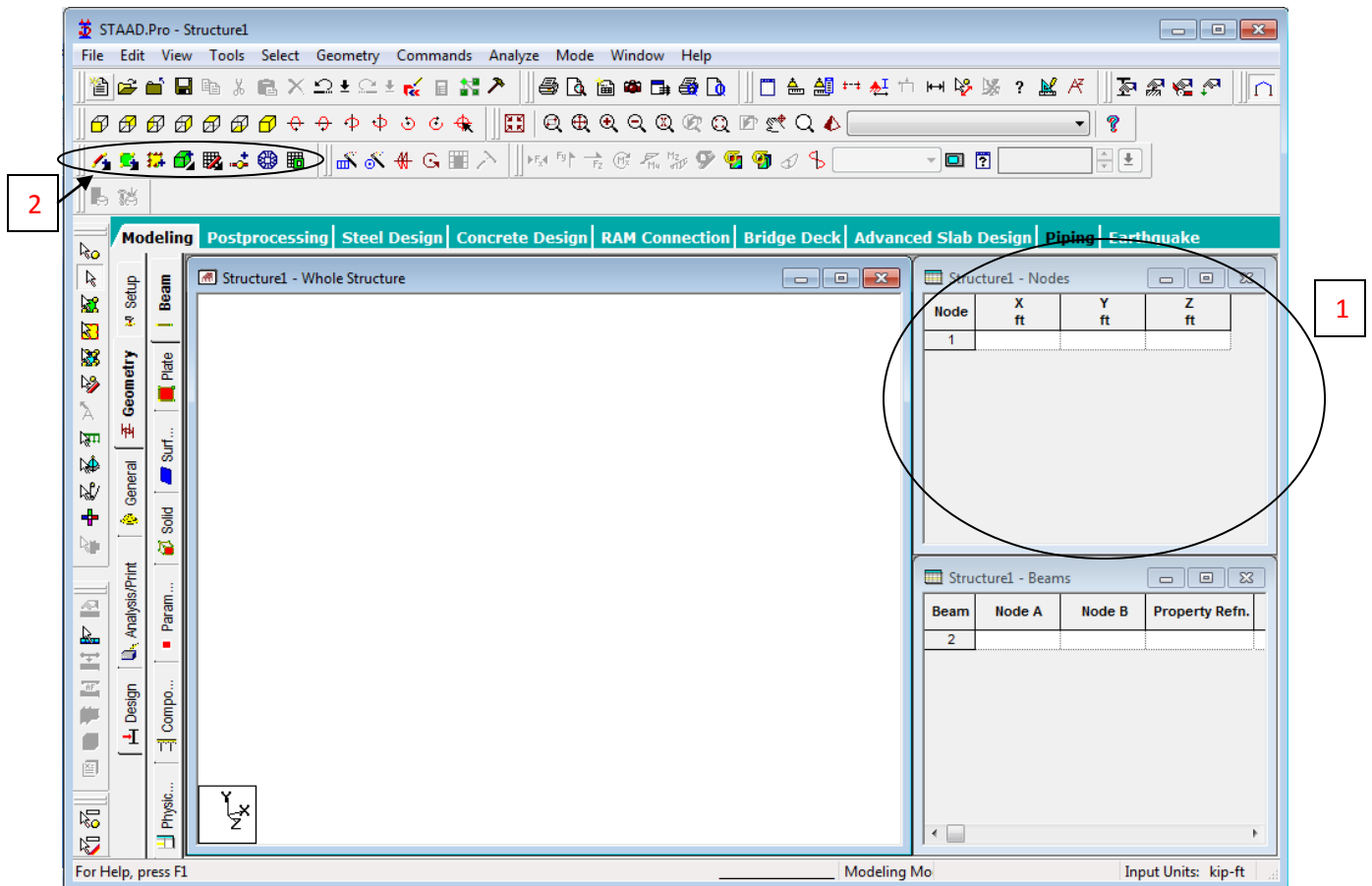
2. Change File Location – It's recommended to create a folder for each model you create because STAAD generates many files

- Graphical User Interface (GUI)



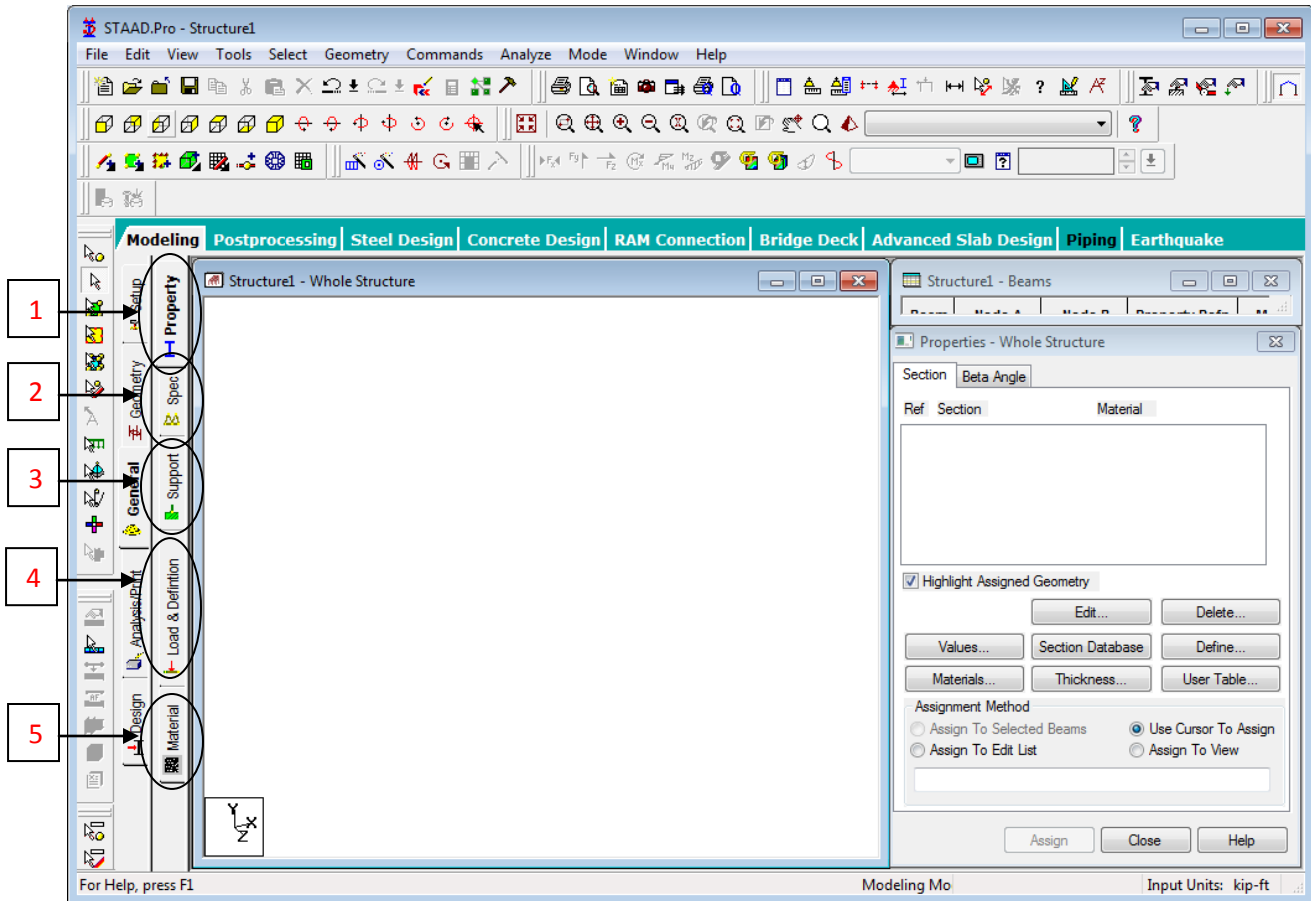
- This area allows for manipulation of the model view
 - The yellow isometric cubes change the face you view from. The other buttons rotate the model. (You can also rotate by right-clicking and dragging)
- Selection Cursor – Use to change between nodes cursor, beams cursor, plates cursor etc.
- Page Control – The organization of the pages, from top to bottom, represents the logical sequence of operations
- Main Window – where the model drawings and results are displayed
- Data Area - where different dialog boxes, tables, list boxes, etc. appear depending on the type of operation you are performing

- Geometry Tab



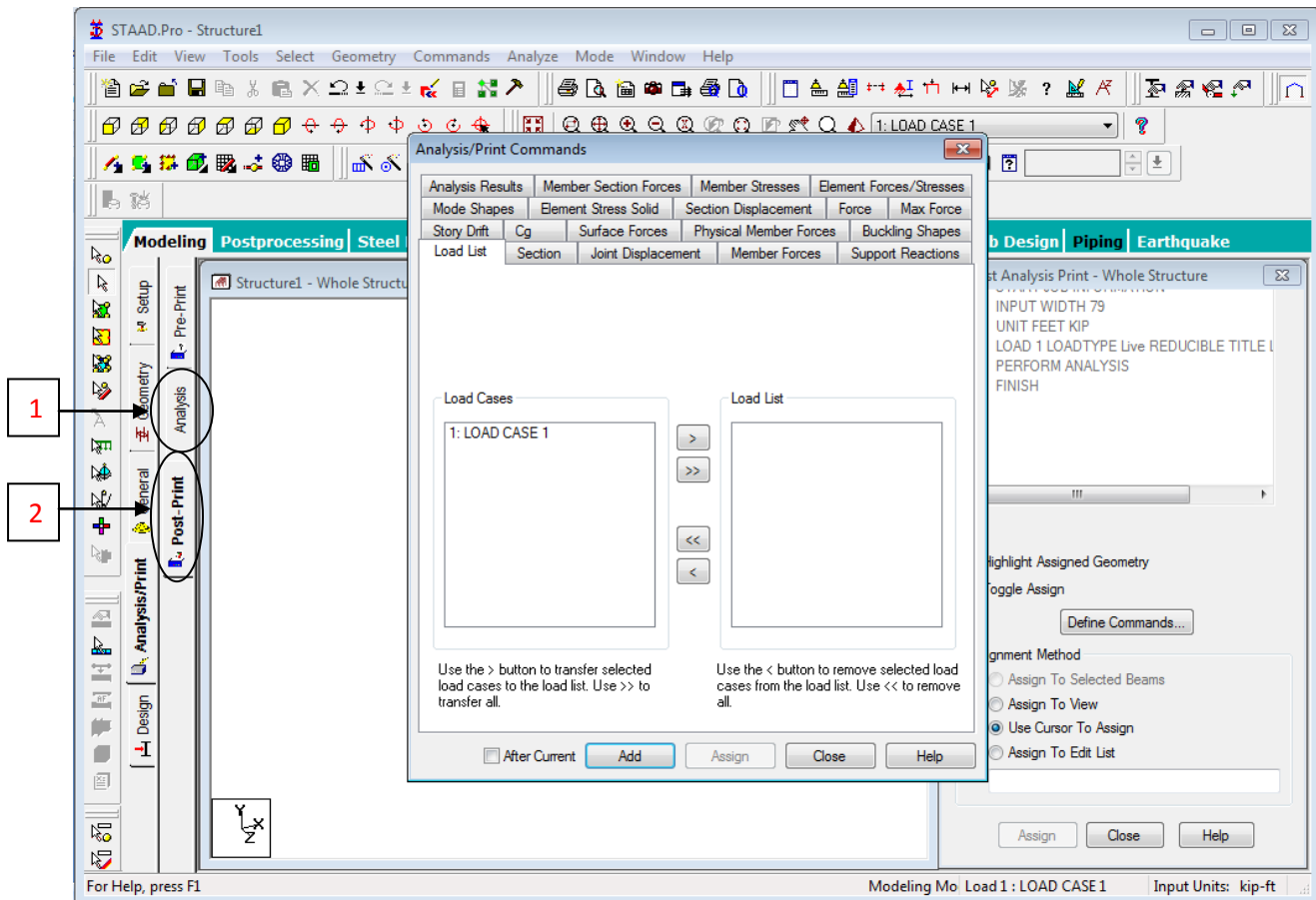
1. Insert nodes manually by defining the X, Y, Z coordinates
2. Drawing tools
 - a. Add beams/plates/surfaces/solids – adds an element using existing nodes
 - b. Snap node/beam – Toggles the drawing grid
 - c. Insert node – insert nodes on existing elements
 - d. Generate surface mesh – creates several nodes on a surface for further analysis

- General Tab



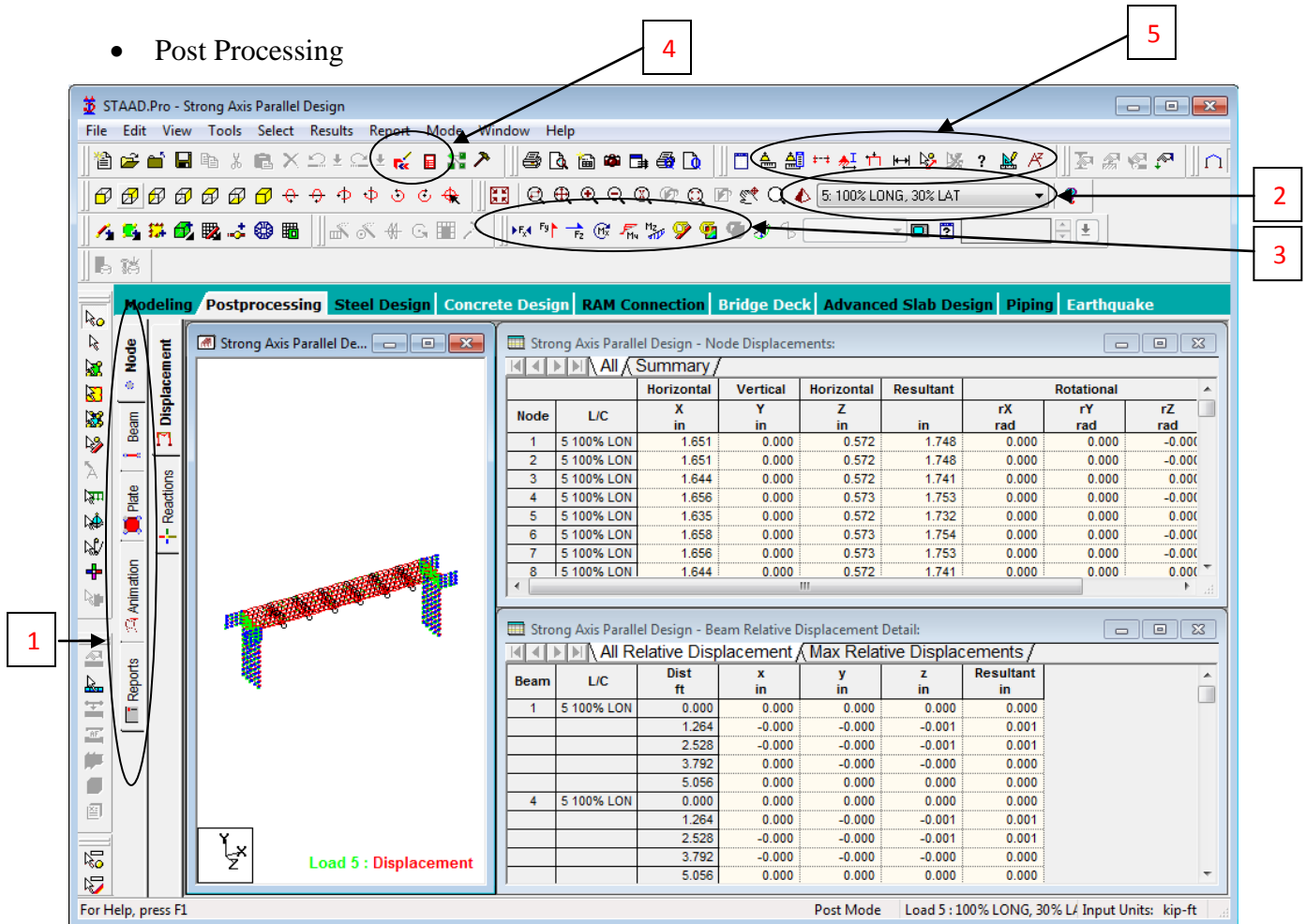
1. Property – Define properties for the elements you create (cross-sections, W shapes, plate thicknesses, etc.), also create beta angles (element rotation),
2. Spec – Define member releases, offsets, truss members
3. Support – Define supports for the structure: pinned, fixed, rollers, linear/nonlinear springs
4. Load & Definition – Create load cases to define nodal loads, member loads, area loads
 - a. Load generations can be created for moving loads after defining a vehicle
5. Material – Define new materials if structure uses something other than default materials (steel, concrete, aluminum, stainless steel)

- 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
 - a. After adding the commands, assign them to the members you want the results for

- Post Processing



1. Select desired tab for desired results in Data Space. Our uses:
 - a. Node – node displacements, support reactions
 - b. Beam – beam forces (axial/shear/moment), stresses, etc.
 - c. Animation – view the model animation of displacement
 - d. Reports – create a custom report with desired inputs/outputs for desired load cases/members/nodes
2. The pyramid is the 3D render view and the drop-down menu is the load case selection
3. Turn on shear/moment diagrams to view in the Main Window
4. (Left) STAAD Editor for manual coding. (Right) Output File
5. Unit adjustment, symbols/labels, visual loads, dimension, scale