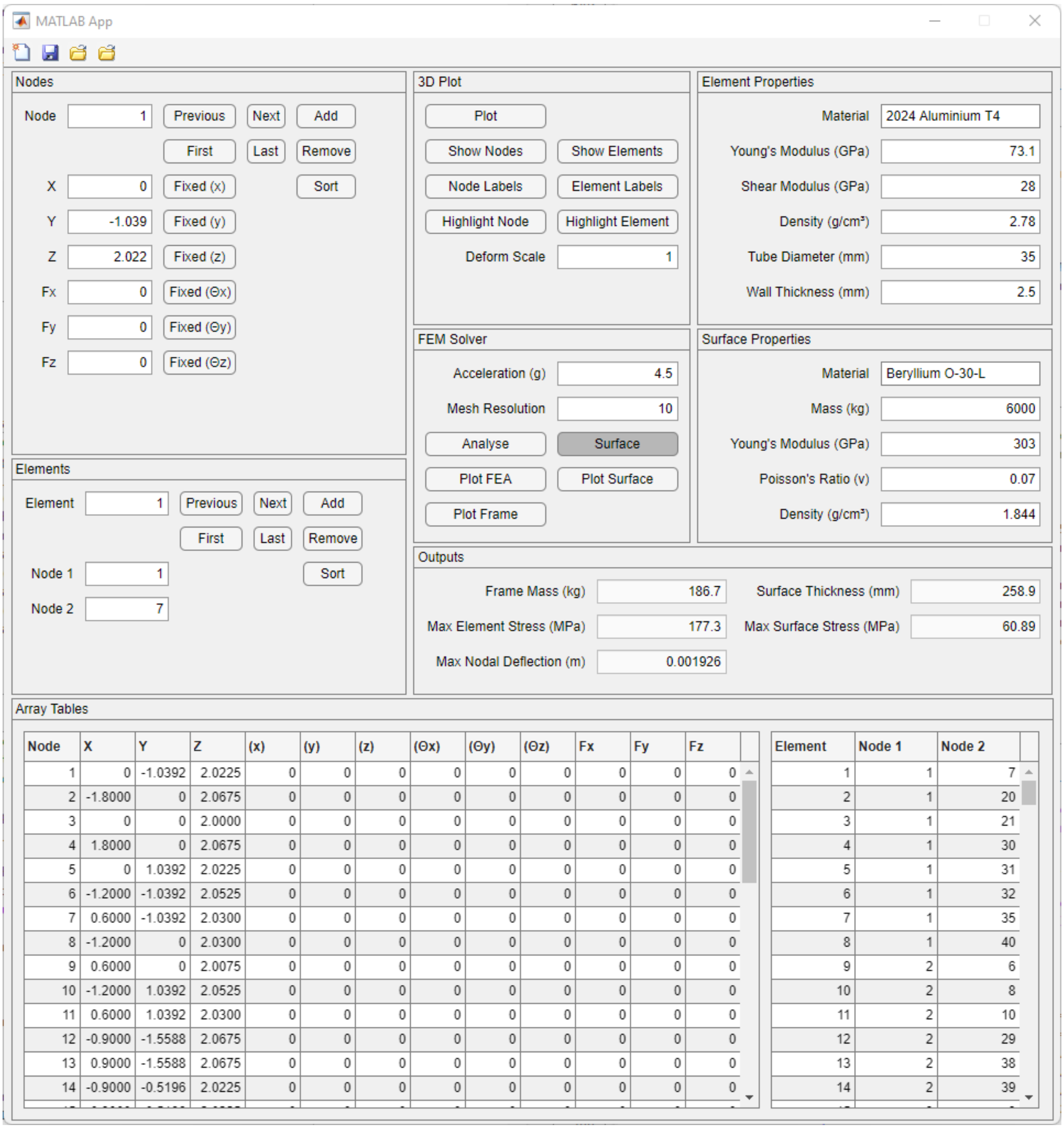
Space Frame Designer Instructions



Updated 02/04/22

Contents

Toolbar

New Space Frame

Clears current session variable, nodes, and arrays.

Save Space Frame

Saves current session to a sub-folder in the current directory, named “Savefiles”.

Load Space Frame

Loads saved session from the folder “Savefiles”. Load file must be formatted according to the app’s save structure.

Import Nodes

Imports a set of nodes into the app’s node array. File must be .mat and formatted with 3 columns corresponding to the x, y, and z coordinates of each node (row).

Nodes

Previous, Next, First, Last

Navigates through the node array according to the node’s number. Does not navigate according to the order in the node array table. Navigating to an existing node will pre-fill the input fields with values corresponding to the selected node.

X, Y, Z, Fx, Fy, Fz

X, Y, and Z set the cartesian coordinates of the node and Fx, Fy, and Fz set the nodal forces along the positive respective axis.

Fixed (x), (y), (z), (Θx), (Θy), (Θz)

Sets the boundary conditions of the node. (x), (y), and (z) fix the node’s coordinates and (Θx), (Θy), and (Θz) fix the node’s rotation.

Add, Remove, Sort

Adding to an existing node will overwrite it with any values in the input fields, including the boundary condition inputs. Remove will remove the highlighted node number from the node array and the element array. In the element array this value will be replace with a zero, and other functions will wait until these “zero nodes” have been removed from the element array. Sort will renumber the nodes according to their magnitude in the coordinate system, priority x, y, z. Useful for removing gaps in the node number system after removing nodes.

Elements

Previous, Next, First, Last

Navigates through element array according to the element’s number. Does not navigate according to the order in the element array table. Navigating to an existing element will pre-fill the input fields with values corresponding to the selected element.

Node 1, Node 2

Defines the start and end node for the current element. Node numbers will change when the node “sort” button is used, this will be accounted for by the software.

Add, Remove, Sort

Adding to an existing element will overwrite it with the node 1 and node 2 numbers. Removing will simply delete the highlighted element from the element array. Sort will renumber the start and end nodes of each element, ascending numerically, and then will sort the elements by start node, ascending numerically.

3D Plot

Plot

Opens a figure containing a 3D space frame constructed of all the primary nodes and elements.

Show Nodes, Show Elements, Node Labels, Element Labels, Highlight Node, Highlight Element

Toggles the visibility of features and labels in the 3D space frame. Highlights the selected node or element number in red in the 3D space frame.

FEM Solver

Acceleration

Multiplies the acceleration component (9.81x) of the distributed force on each element which is calculated using the volume and density of the given element material.

Mesh Resolution

Determines how many elements construct a member in the space frame being solved; 1x is a singular element per member.

Analyse

Runs the Finite Element Analysis on the space frame structure.

Surface, Plot Surface

Prompts an import of a selection of nodes, which if are already present in the space frame will become loaded nodes to support a surface – in this specific build, the code automatically selects just the first 25 nodes as this allows the original coursework node file to be used to specify the surface nodes[1]. Plot surface graphically shows the surface deflection and the surface von Mises stress concentration.

Plot FEA, Plot Frame, Deform Scale

Plot FEA opens another figure showing the deformed space frame structure according to the FEA. Blue members are under tension, red are under compression and green are neutrally loaded. Plot Frame overlays the undeflected space frame for reference. Deform scale multiplies visual deformation to give a clearer view of the weak points. 1x: 1m = 1m.

[1] In the future I would like to refine this clunky feature. My implementation would add another flag to each node, like the boundary condition flags, which would specify a node as a “surface node”.

Improvements

If I had had more time to develop the app, I would have made numerous improvements to the useability, GUI and function.

Changing Boundary Condition Method

Using loading bars, it is clear that the slowest step of the FEA calculation is the boundary condition step. I suspect that this is because I am removing the degrees of freedom by deleting rows of the global stiffness matrix and force vector, causing them to dynamically resize with every loop, reducing performance. Instead, I would use the penalty method, or would add an option to, as this would not change the size of the matrices, reducing the operations needed per degree of freedom.

More Element Analysis

The only quantifiable outputs implemented were maximum element stress, maximum surface stress and maximum nodal deflection. Using the element force vector, it is possible to further analyse the forces on the elements and compare them to different material properties to ensure that the material selected is suitable for the space frame. Further figures plotting deflections and internal element stresses could have been implement which would have provided further insight into the structure's performance.

Full rewrite of the code

Due to the specific requirements of the coursework, certain aspects of the code became too specialised to be used as a general-purpose space frame element FEA solver. Therefore, I would rewrite the input method of the app, allowing for much more flexibility of design, much more streamlined data management and the ability for other element types such as beam or plane frame to be modelled. By dynamically adding flags to each node, it would be possible to only enable certain boundary conditions or separate model sections to be enabled (distributed load acting upon a space frame for example).