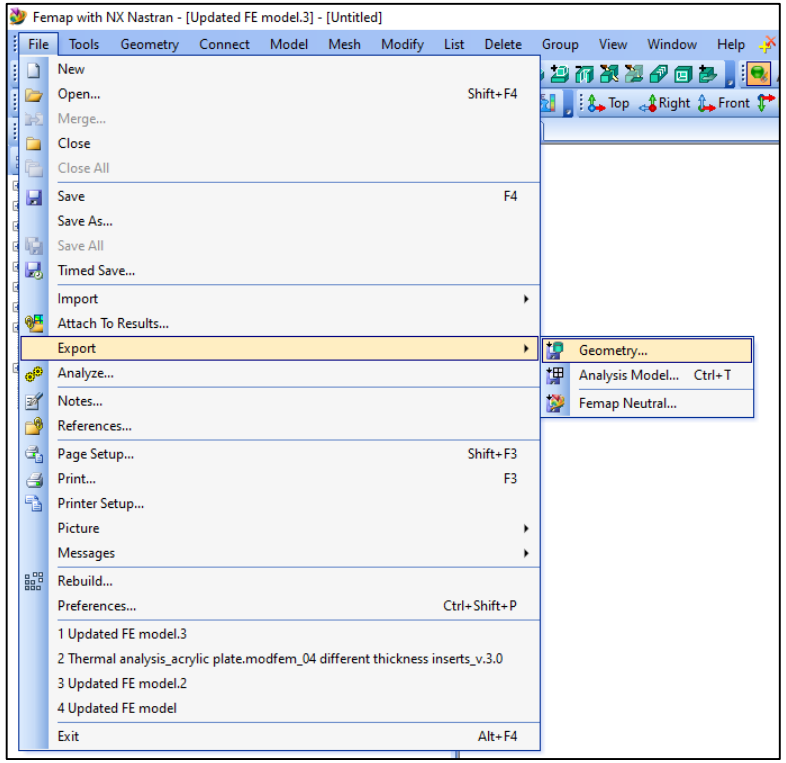
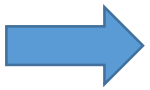
Thermal analysis

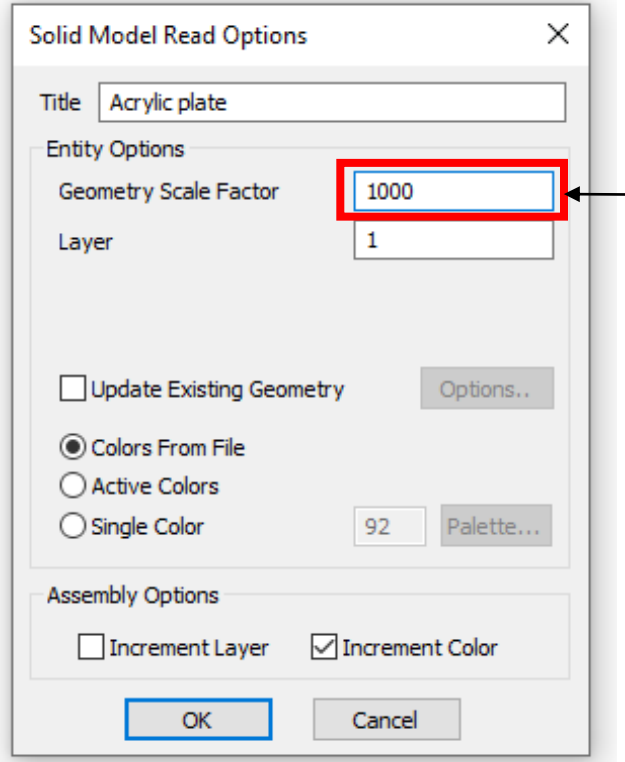
Step by step guide

1. Import geometry:

Make the 3D model of the plate in solid edge and export as parasolid or step file. Import to Femap



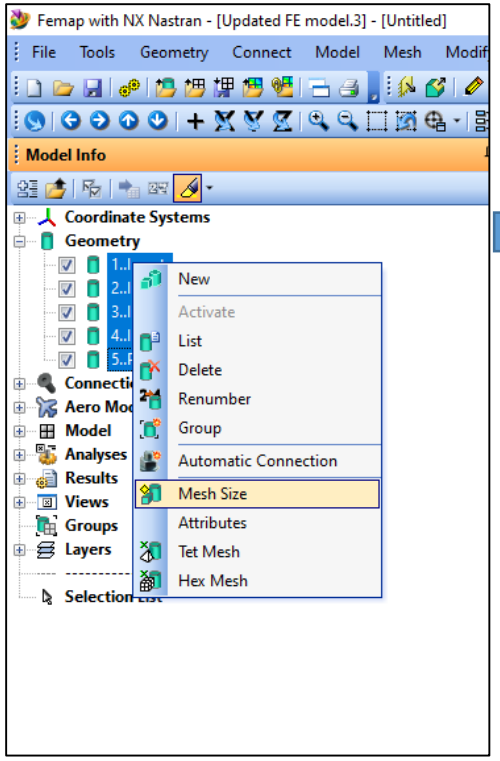


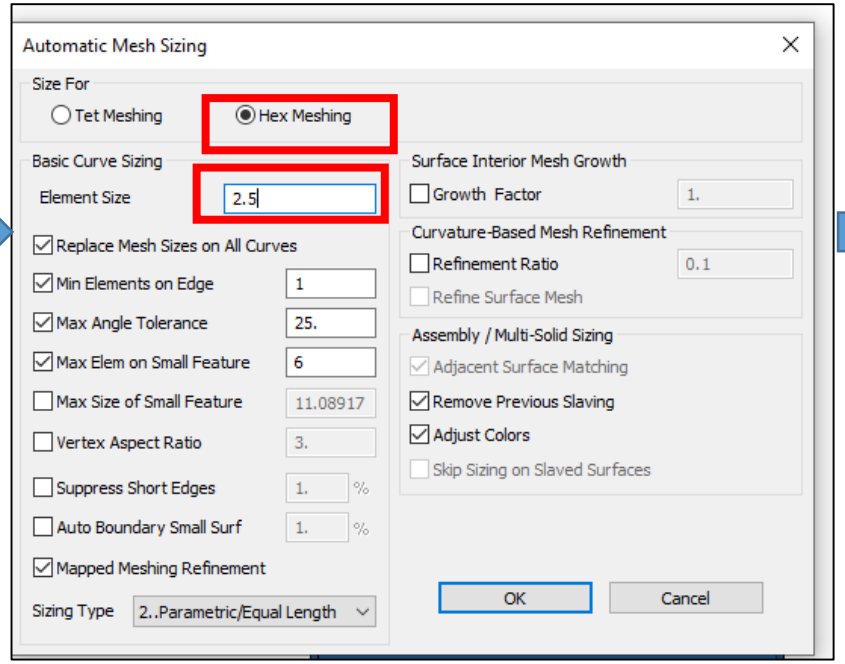


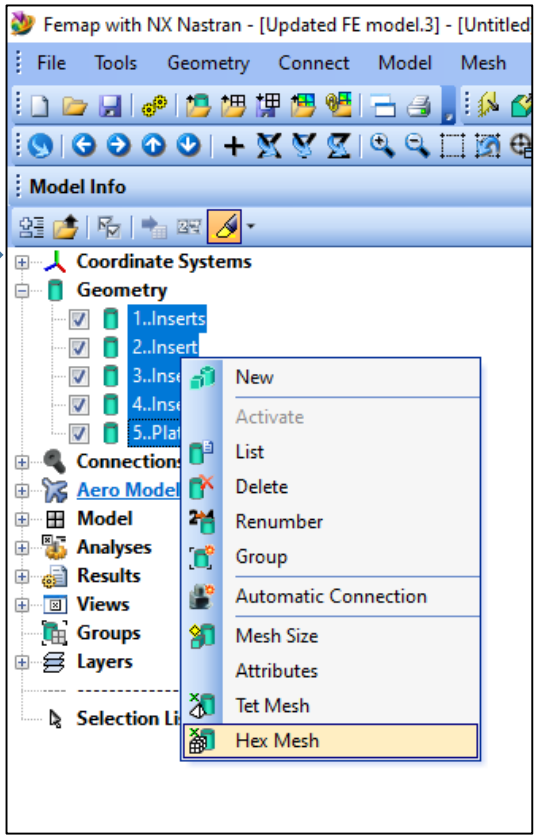
For mm put 1000

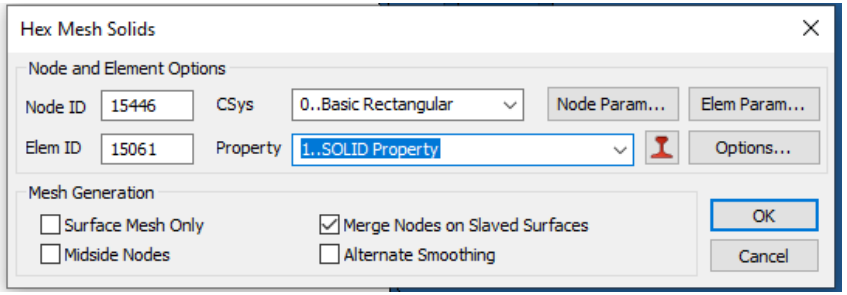
2. Meshing:

Hex mesh is used to meshed the plates and inserts because hex mesh provide accurate results.

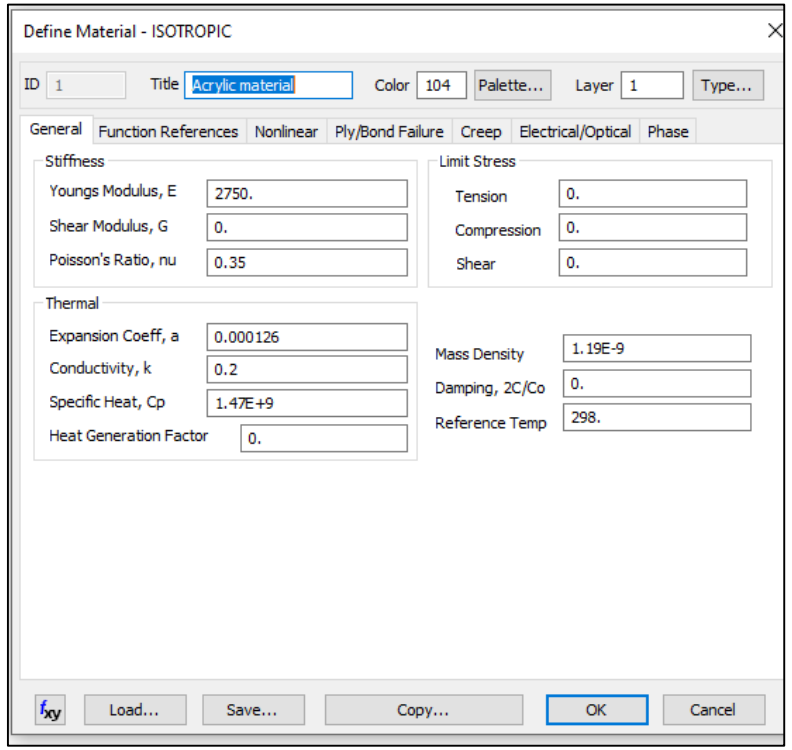






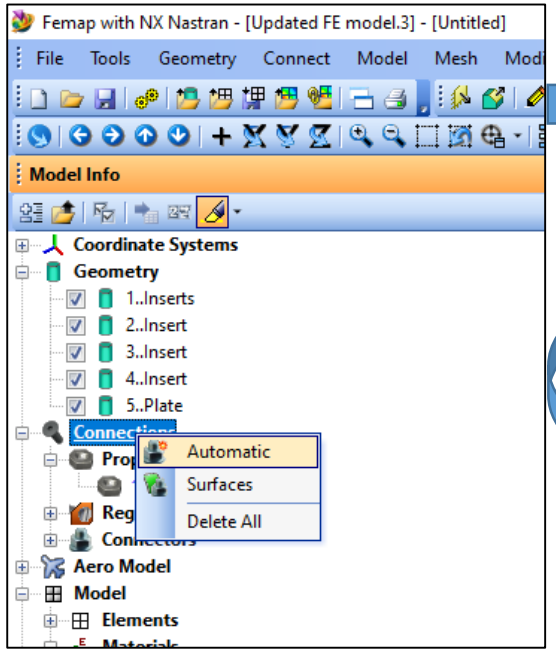


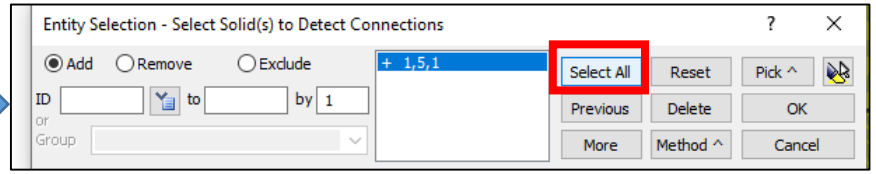
3. Define material:

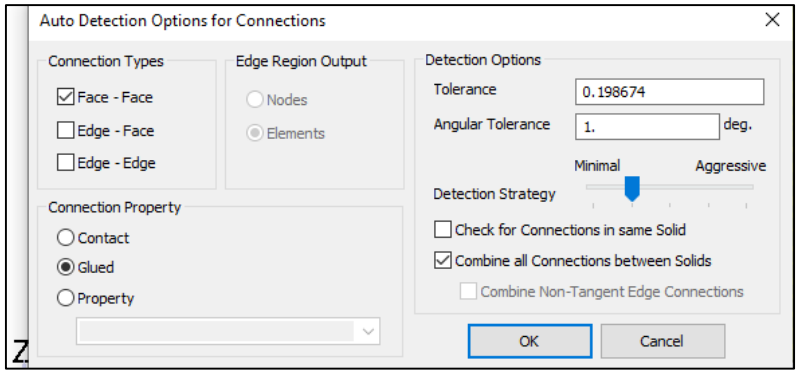


4. Glue connection:

Glue property defined to make connections between the inserts and plate

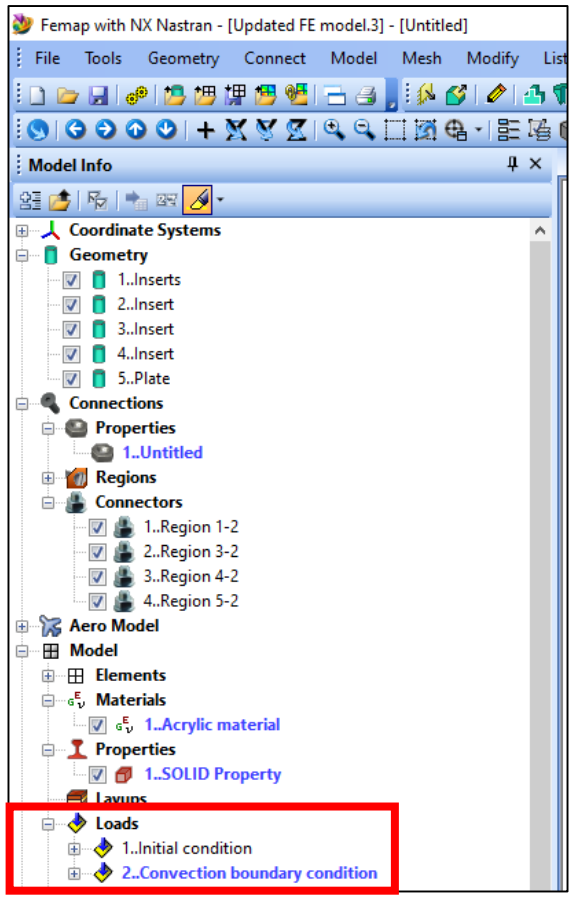


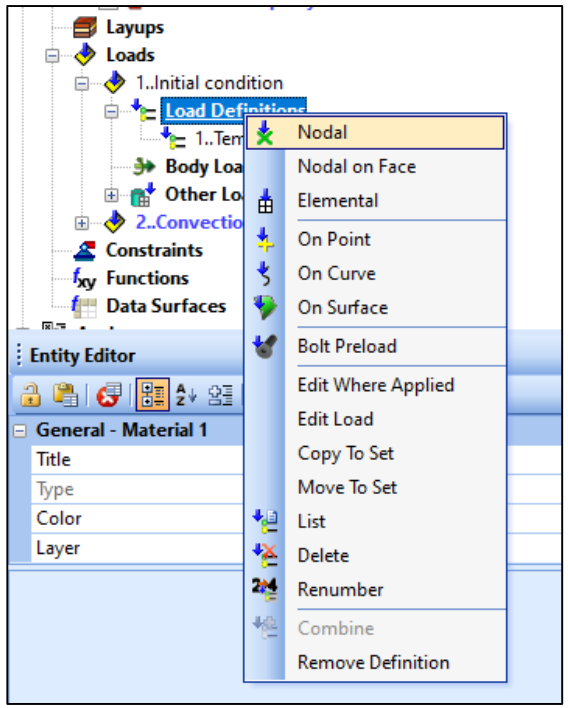


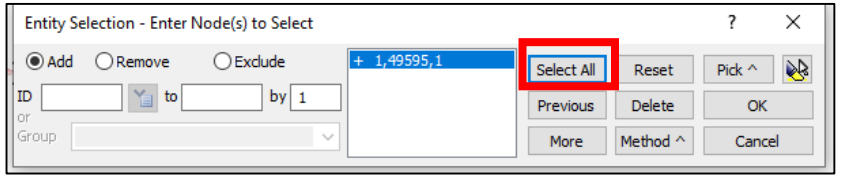


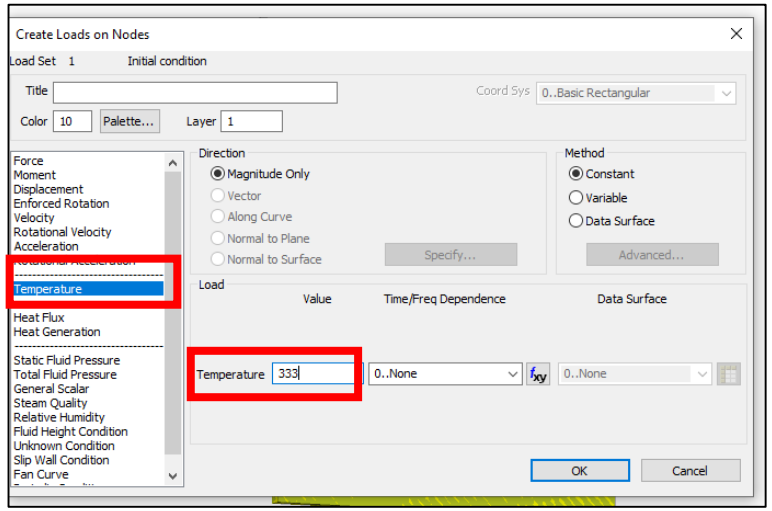
5. Load:

Two load set are defined for convectional analysis **Initial condition**

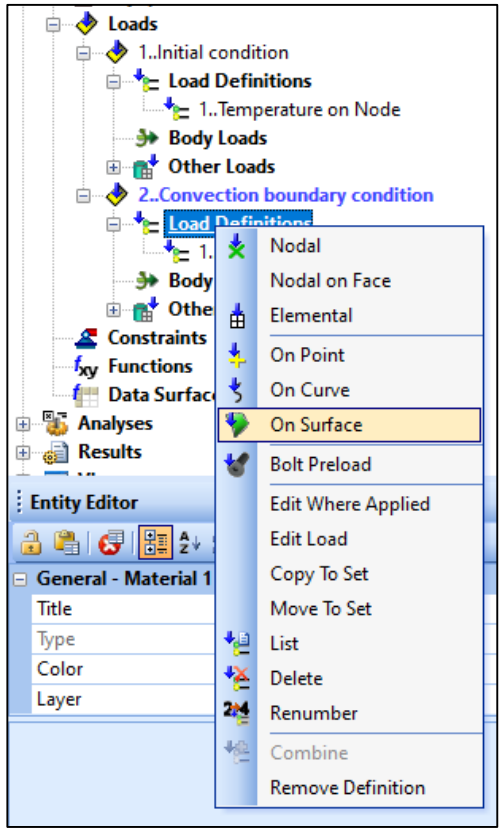


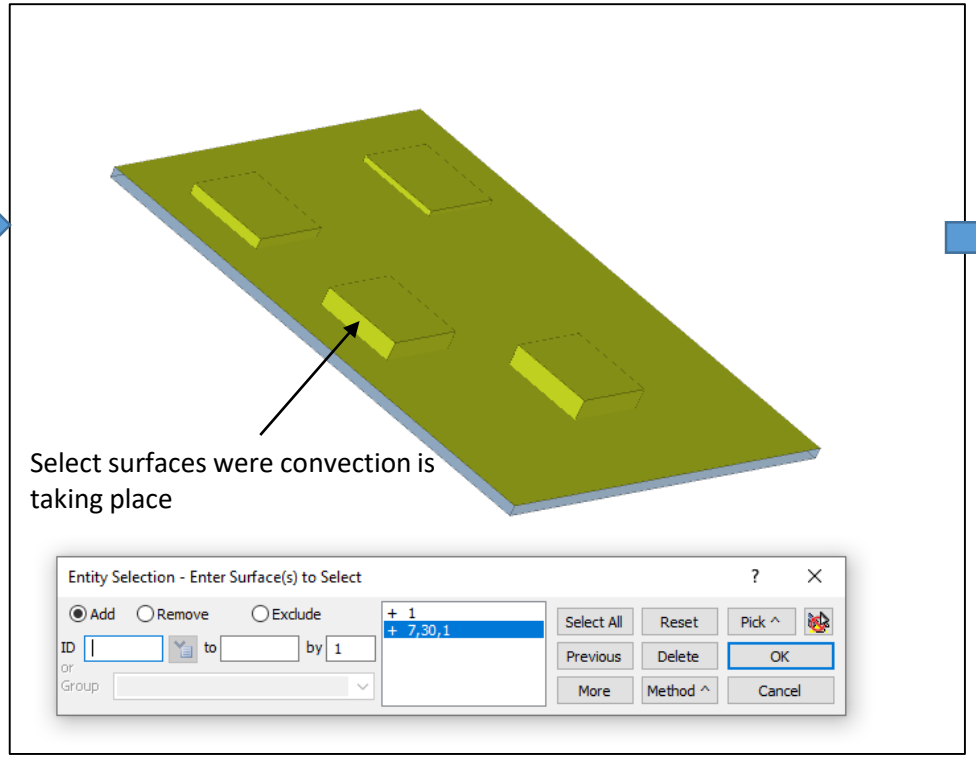


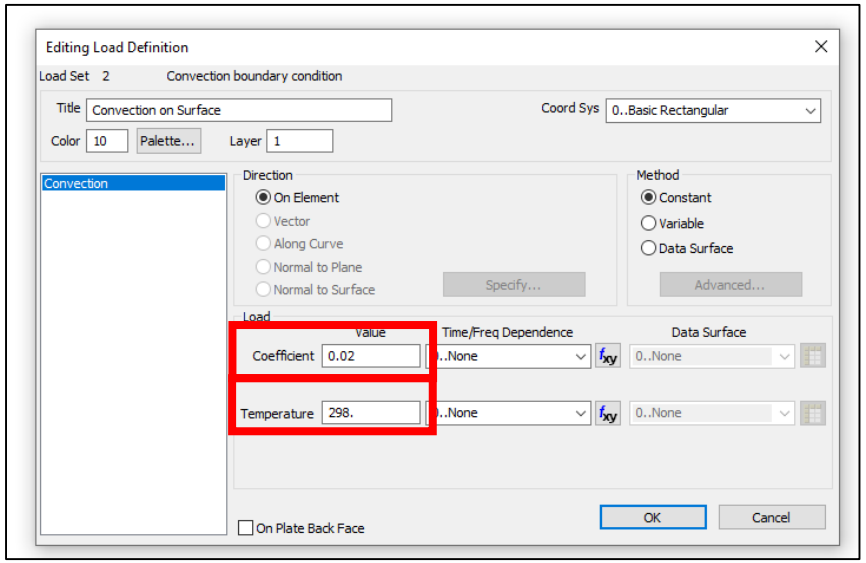




Convection boundary condition

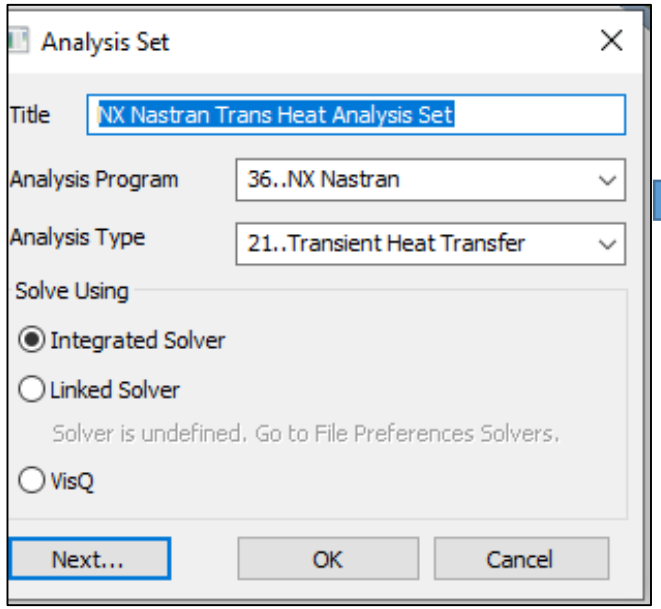


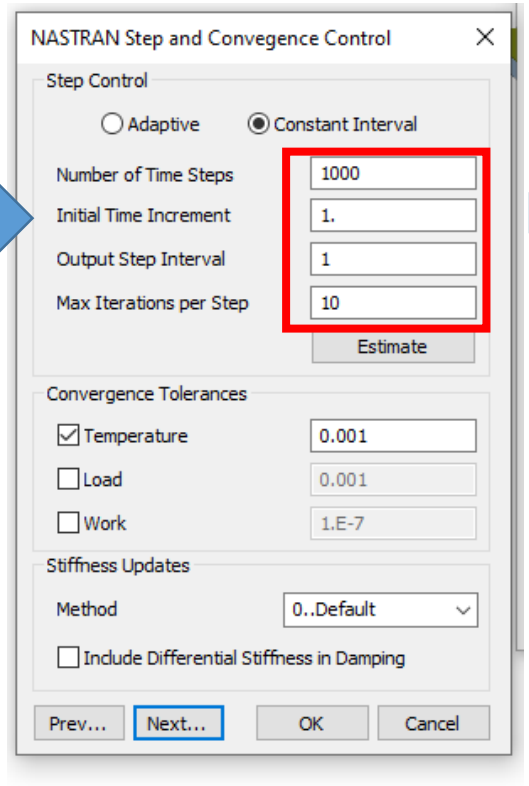


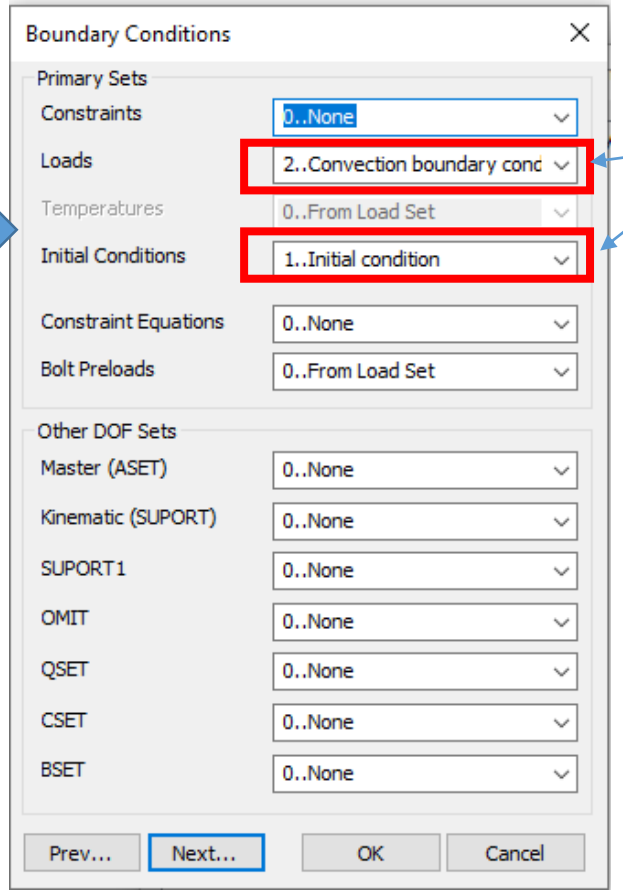


6. Setup analysis:

Keep the default values in all windows in the analysis setup except the following windows



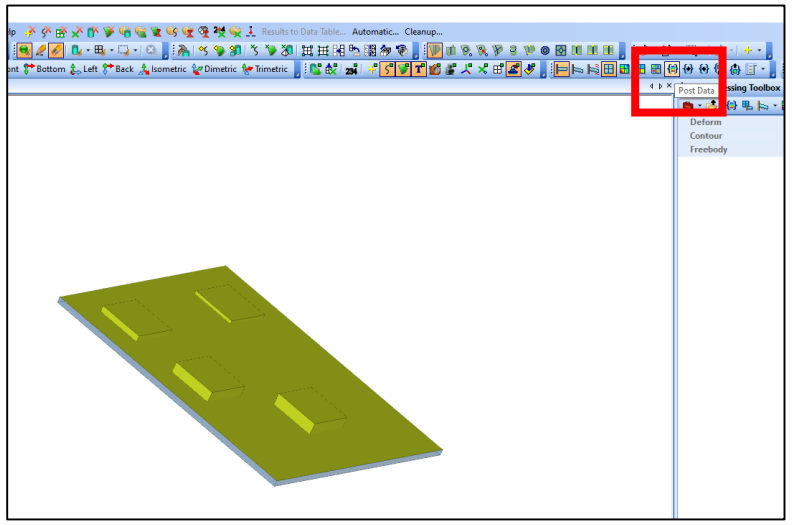


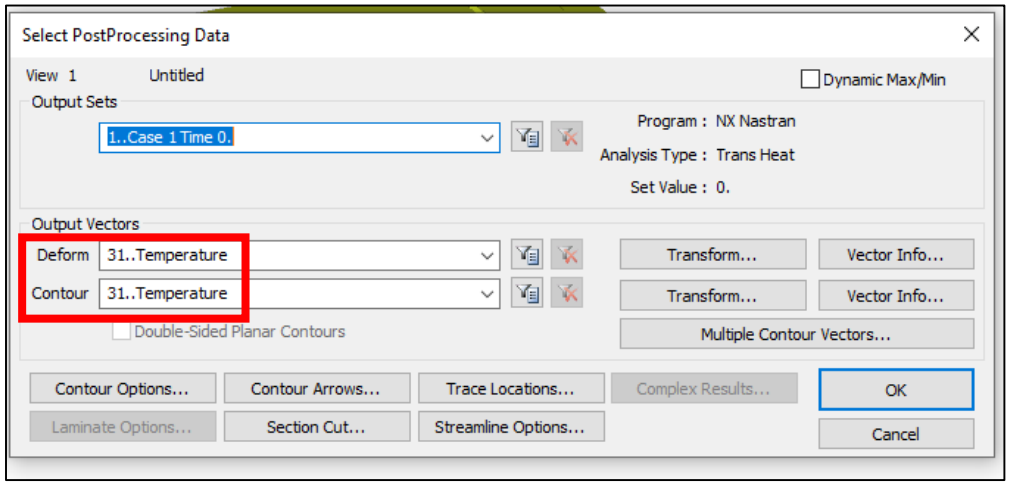


Put the load case in relevant box

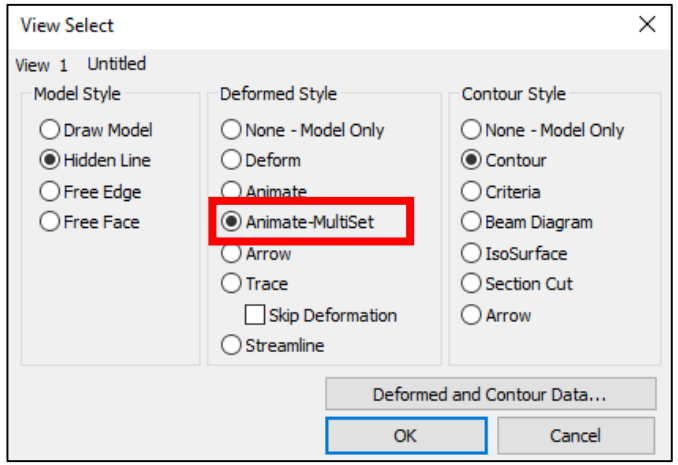
7. Postprocess:

Animation of the temperature.



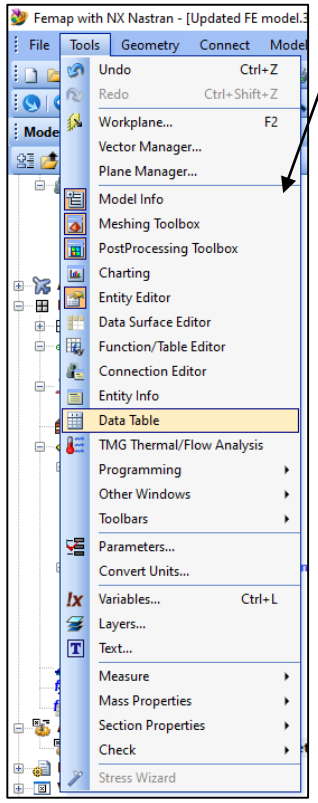


Press F5 to get the view select windnow

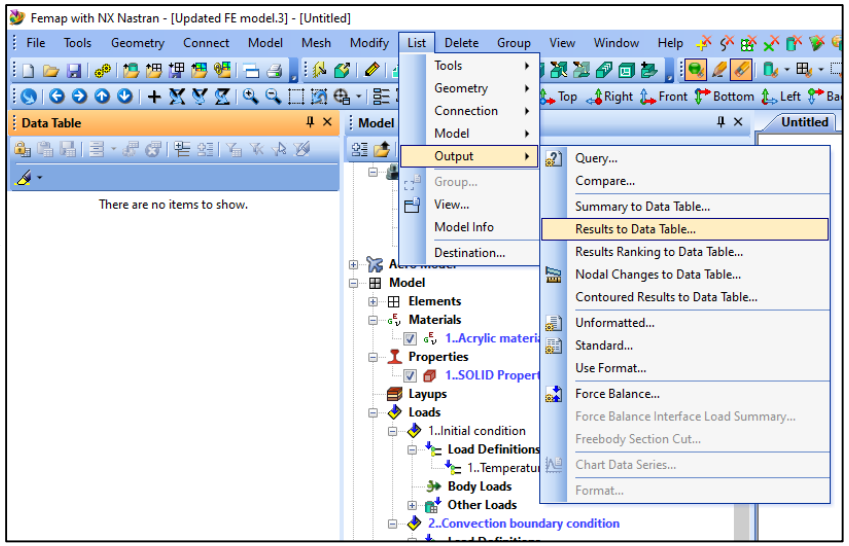


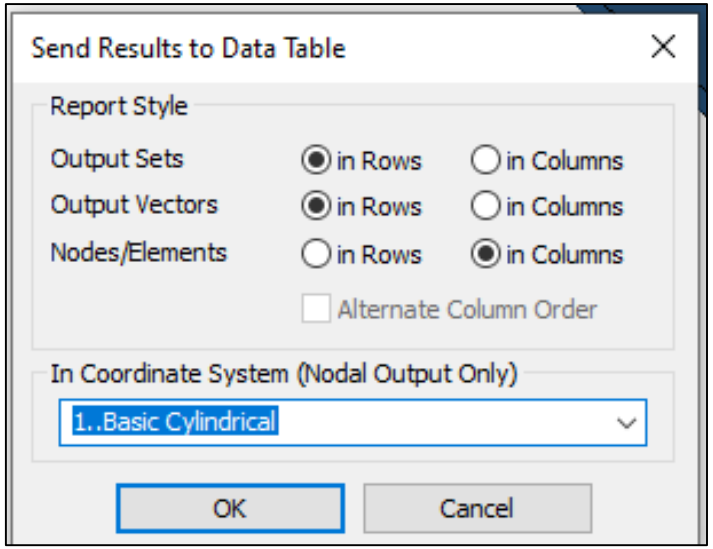
8. Data table:

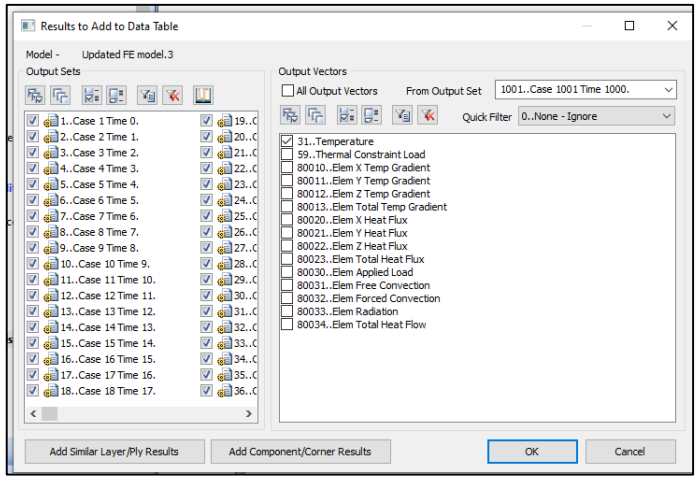
Extract temp on nodes of the plain surface of the plate.

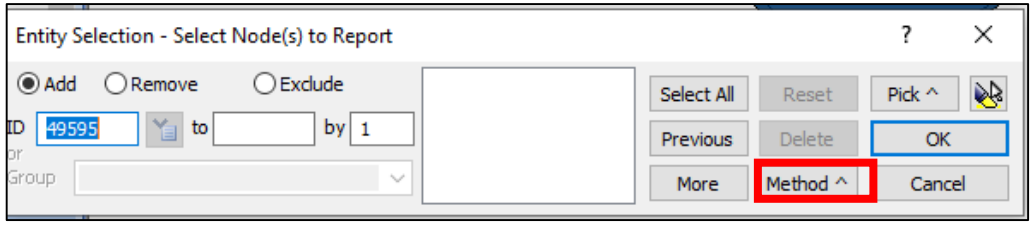


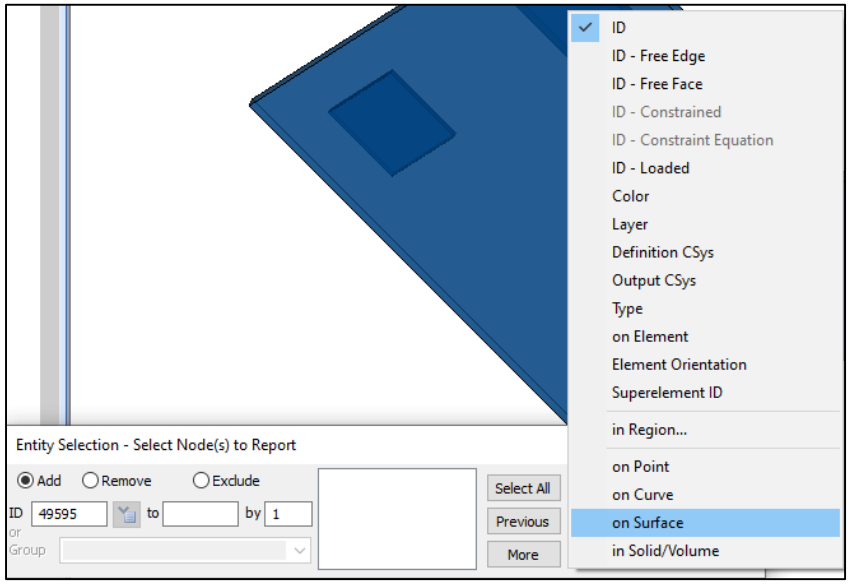
Activate data table

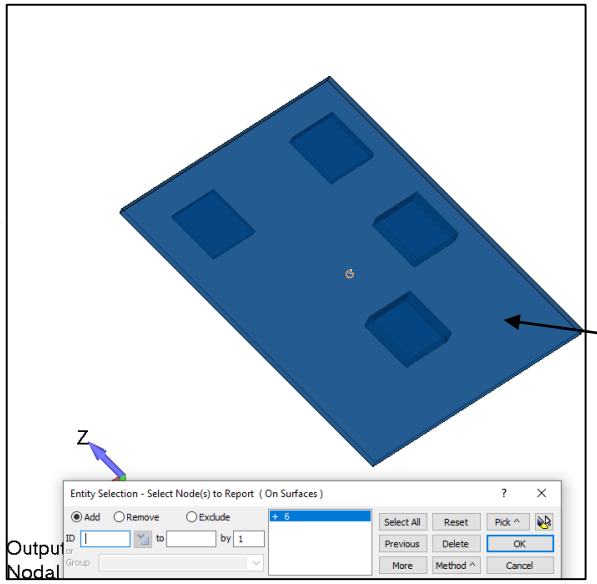




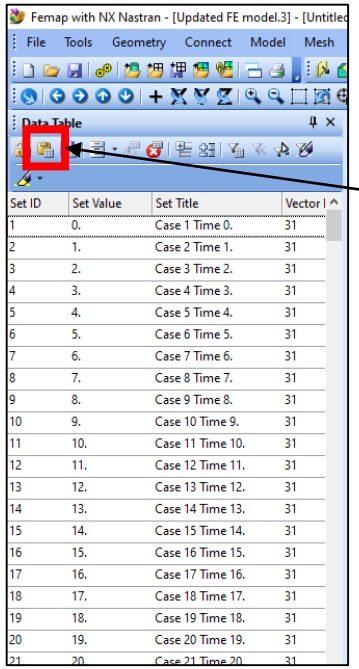








Select plain surface



Set ID	Set Value	Set Title	Vector
1	0.	Case 1 Time 0.	31
2	1.	Case 2 Time 1.	31
3	2.	Case 3 Time 2.	31
4	3.	Case 4 Time 3.	31
5	4.	Case 5 Time 4.	31
6	5.	Case 6 Time 5.	31
7	6.	Case 7 Time 6.	31
8	7.	Case 8 Time 7.	31
9	8.	Case 9 Time 8.	31
10	9.	Case 10 Time 9.	31
11	10.	Case 11 Time 10.	31
12	11.	Case 12 Time 11.	31
13	12.	Case 13 Time 12.	31
14	13.	Case 14 Time 13.	31
15	14.	Case 15 Time 14.	31
16	15.	Case 16 Time 15.	31
17	16.	Case 17 Time 16.	31
18	17.	Case 18 Time 17.	31
19	18.	Case 19 Time 18.	31
20	19.	Case 20 Time 19.	31
21	20.	Case 21 Time 20.	31

Copy temperature
from here and past
in the spread sheat

If you have any question please feel free to ask