Step 1: Create Database

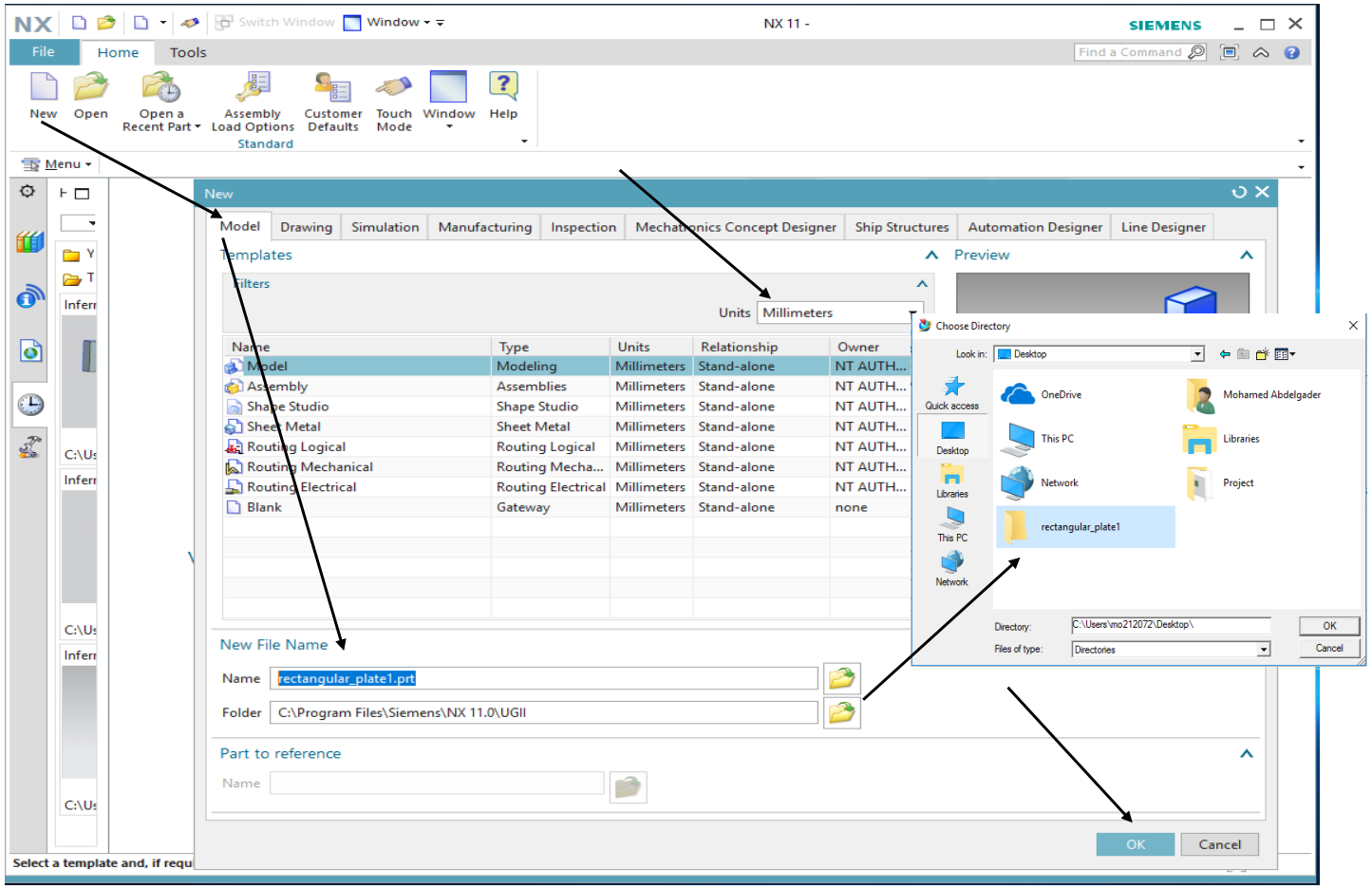
a) **File-New**

b) Choose **Model** from New options and change units to **inches**

c) Then change **New File Name** to be “rectangular_plate1”

d) Make a new folder on the desktop under “rectangular_plate1”

e) Then click **Ok**

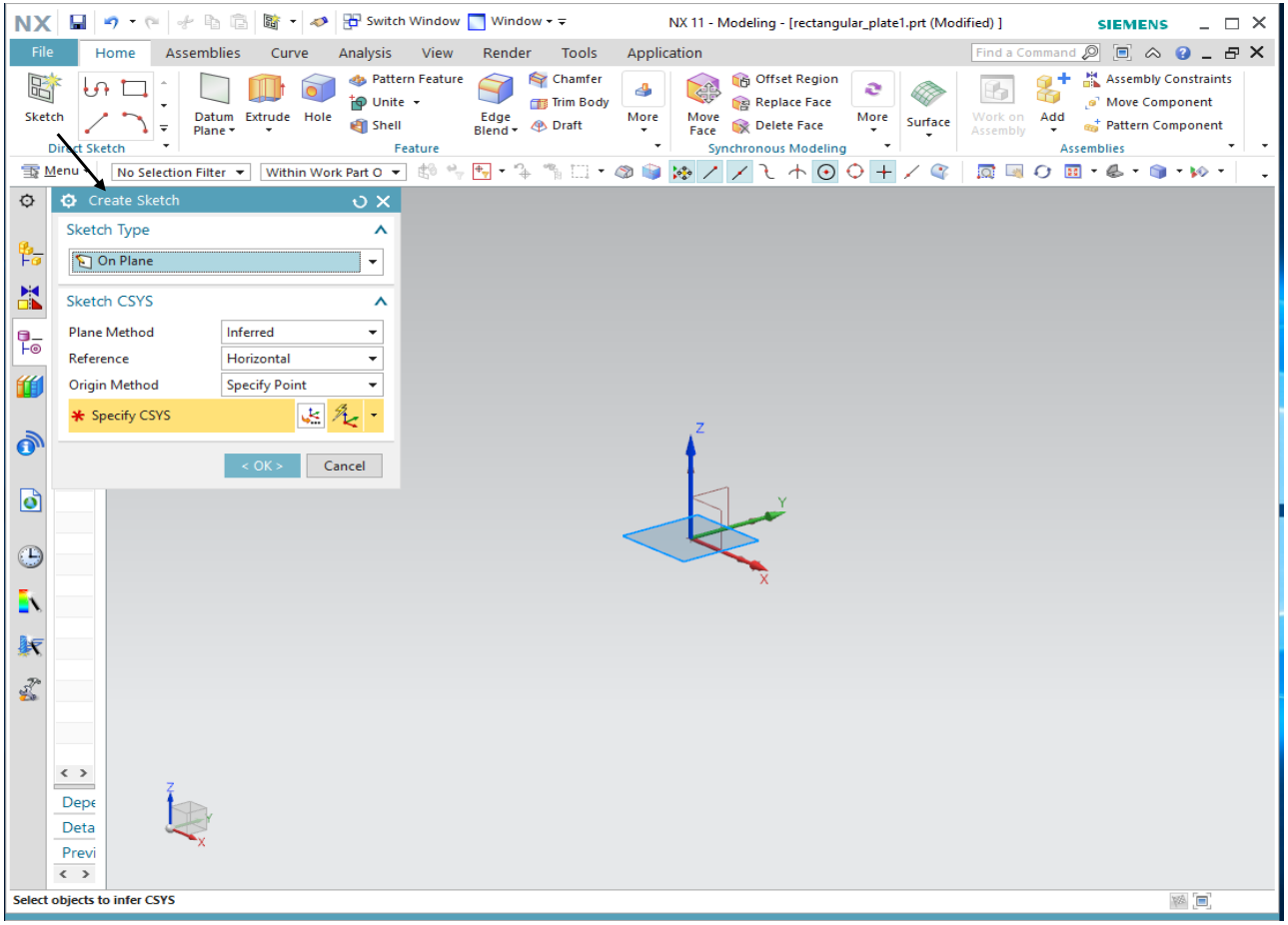


Step 2: Create a rectangular surface

a) **Create a sketch**

b) Sketch type **on plane**

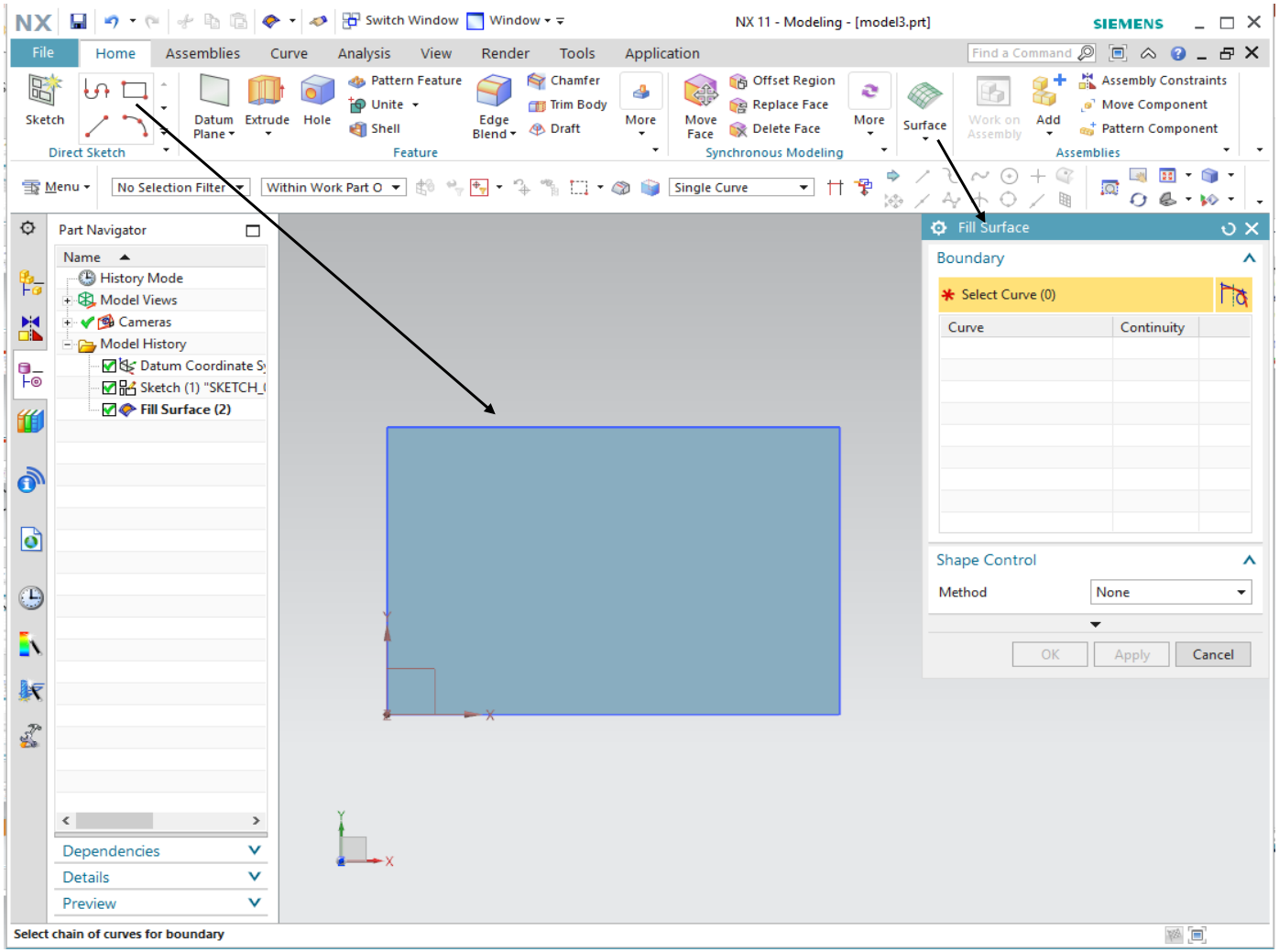
c) Then click **Ok**



d) **Create a rectangular (12 in *8 in** as we can see on the screen *(tips: scroll down to zoom out so it fits on the window, F8 if you lost the plane view)*

e) Then go to **Surface** in the home page, choose **Fill surface,** make sure all the sides of the rectangle are selected **,** then click **ok**

f) Save the file by clicking **Control S.**



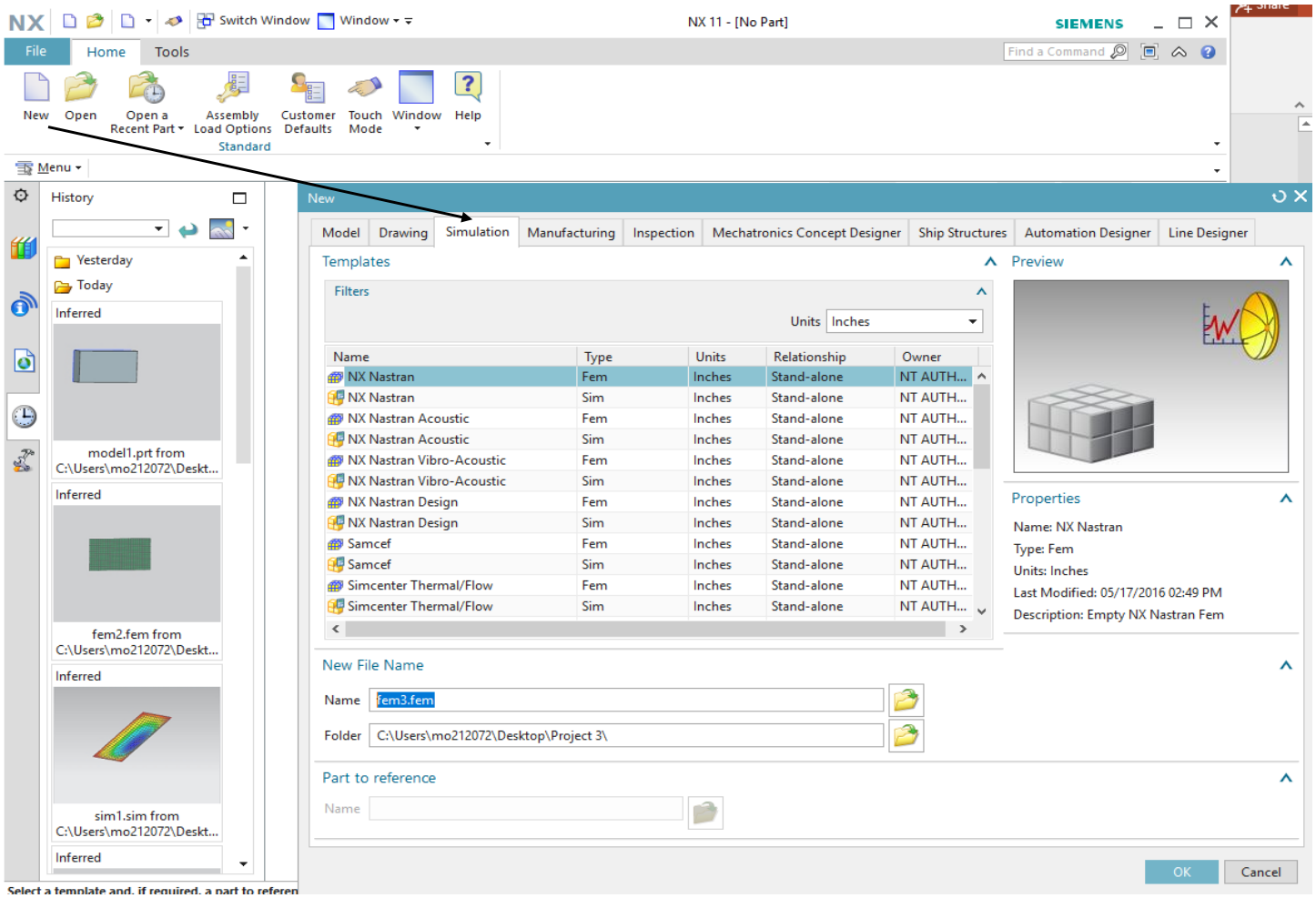
Step 2: Mesh the surface

a) **File-New**

b) Choose **Simulation** from New options

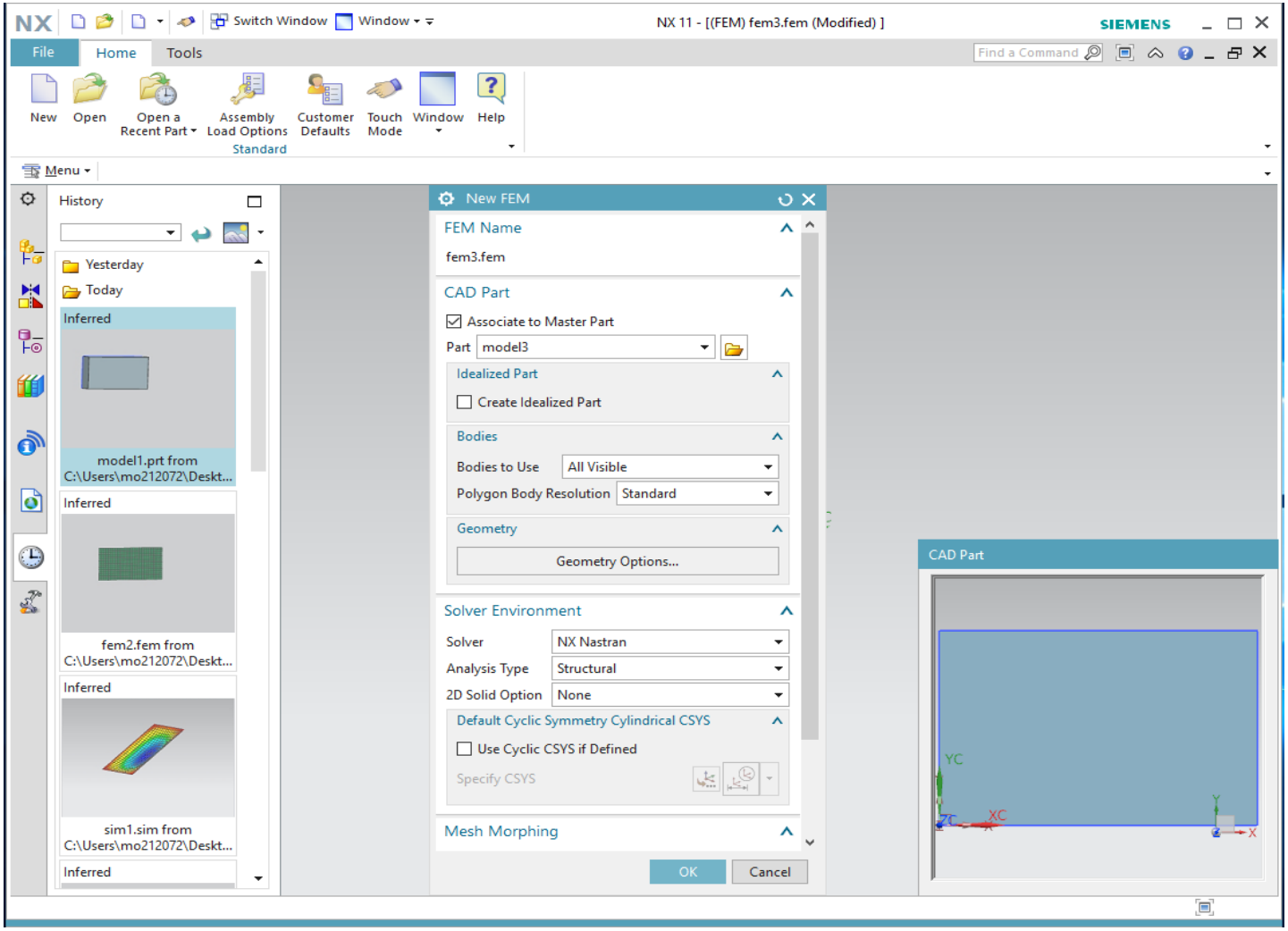
c) Then choose NX Nastran **Fem Type**

d) Then click **Ok**



a) You will see this window

b) Just click **Ok**

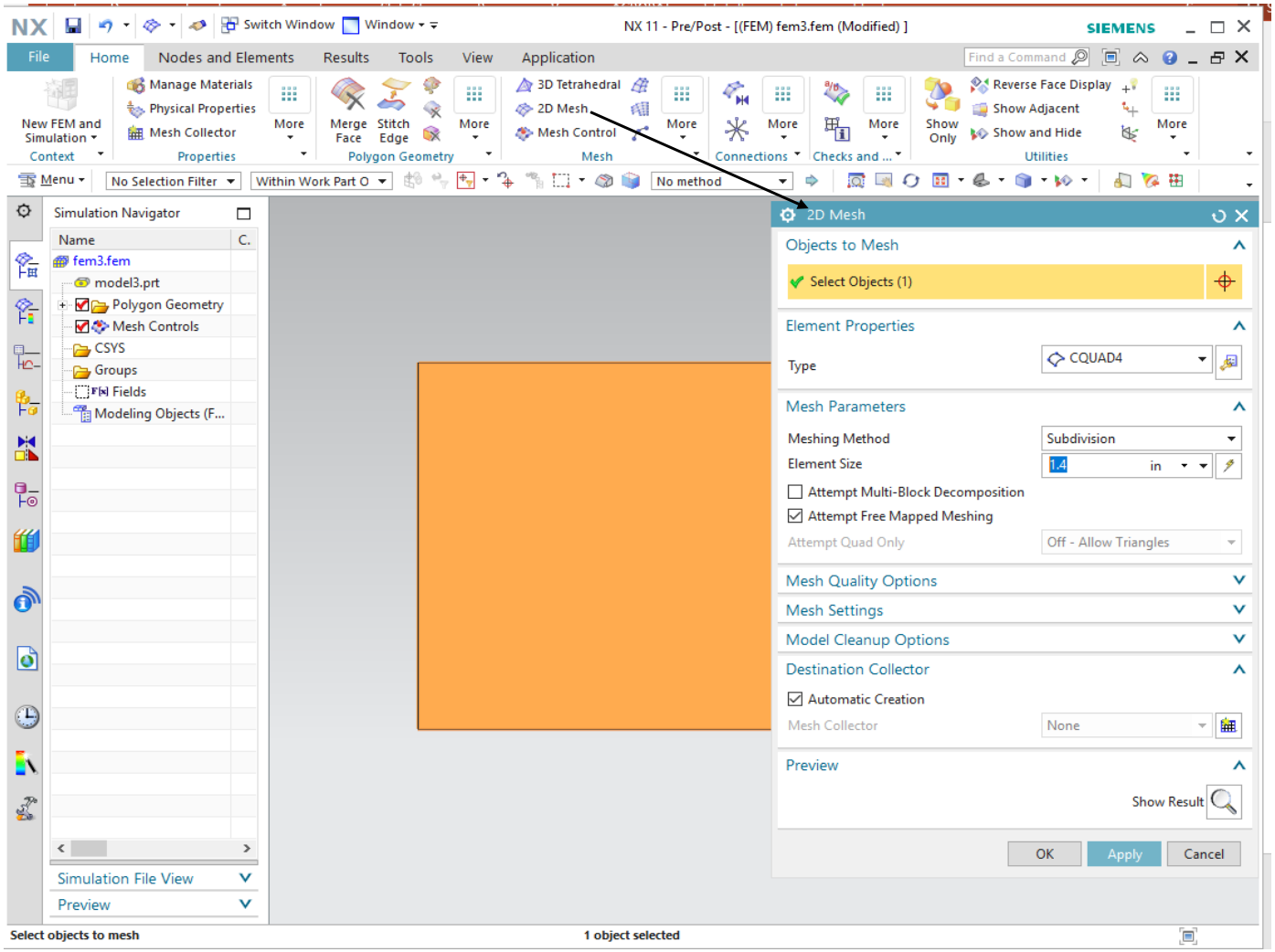


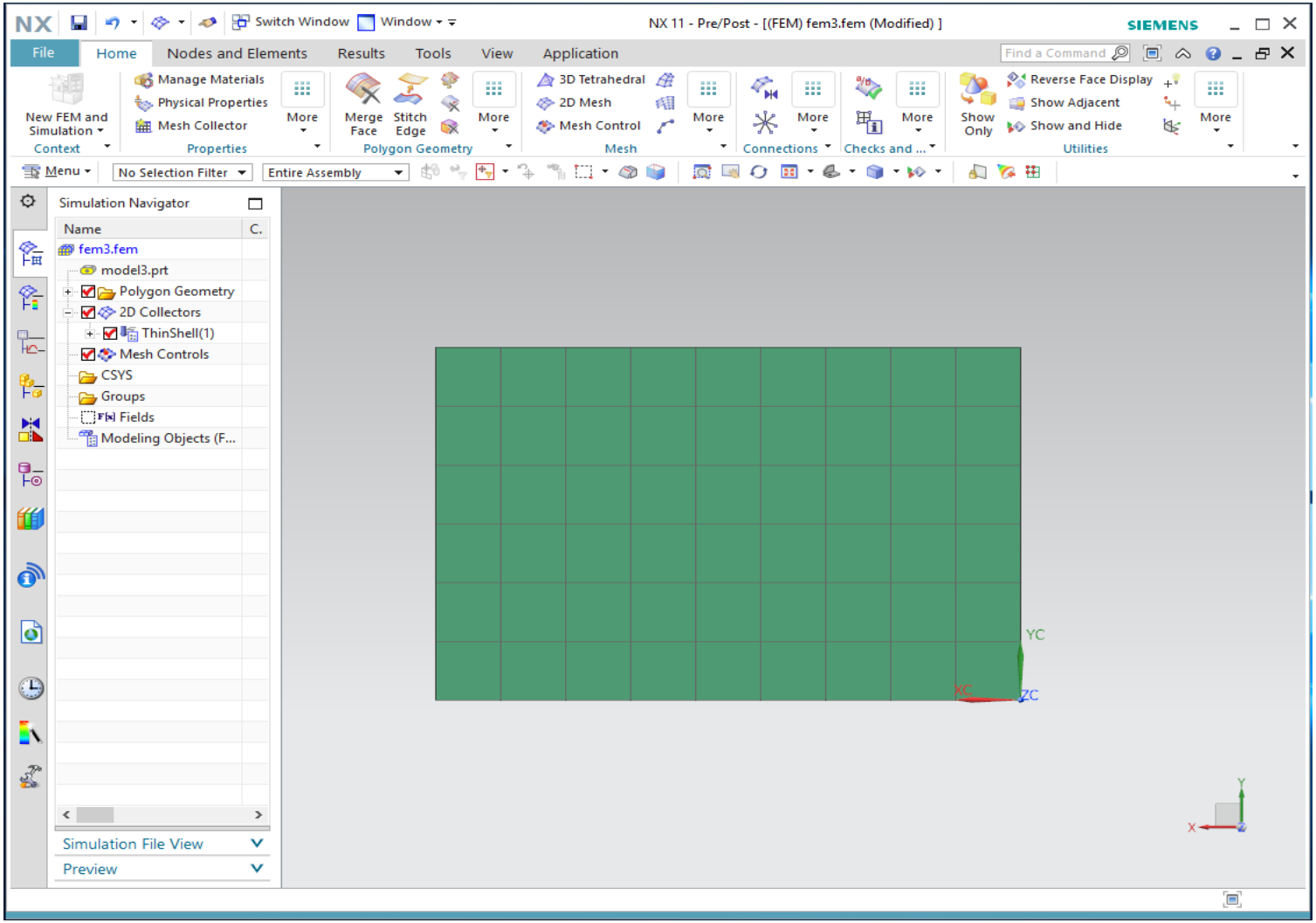
a) **2D Mesh** from the home menu and select the surface of the object (Polygon face)

b) Element type **CQUAD4**

c) Element size **1.4 in**

d) Then click **Ok**



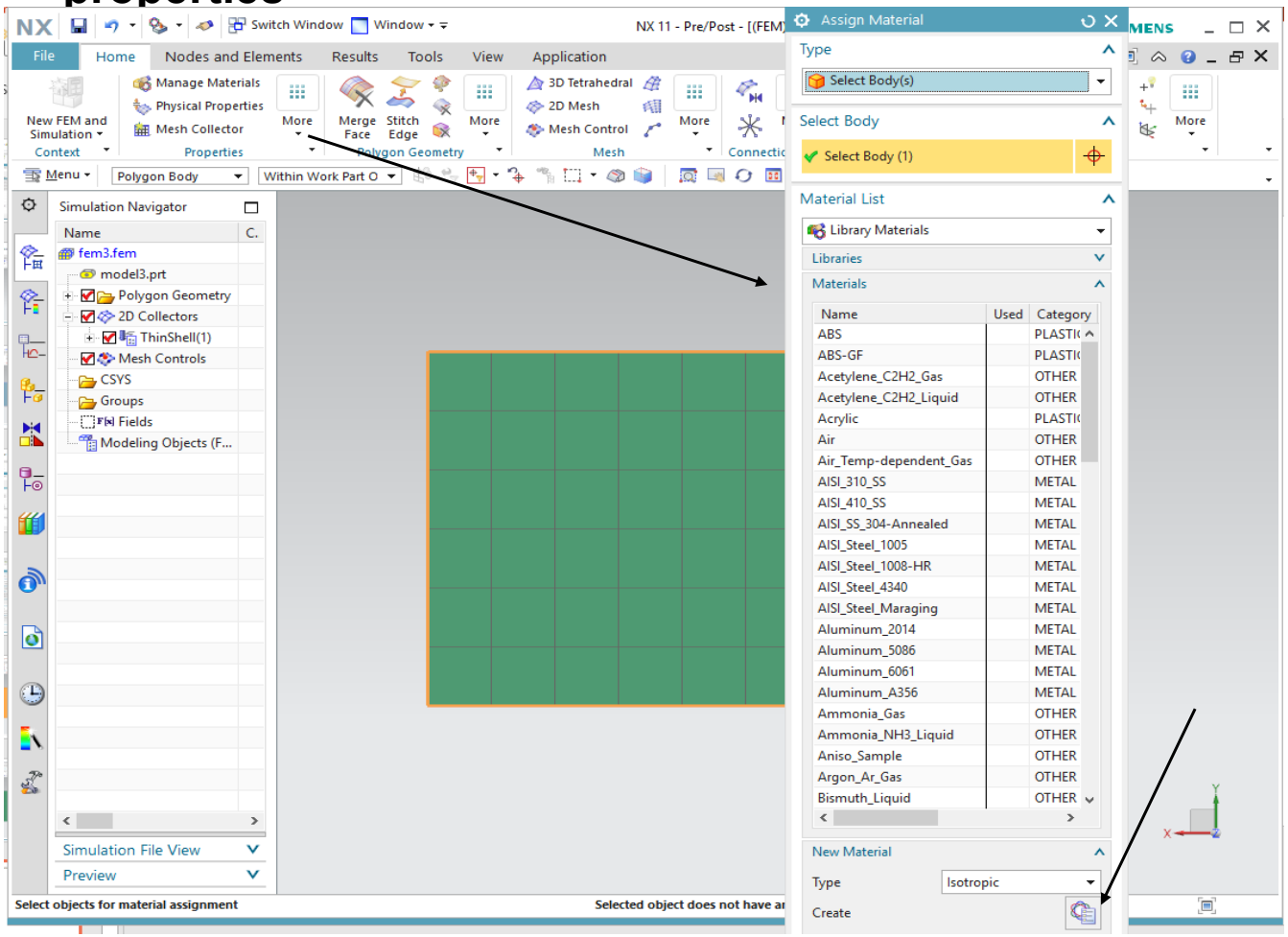

NX
Switch Window
Window
NX 11 - Pre/Post - [(FEM) fem3.fem (Modified)]
SIEMENS
File
Home
Nodes and Elements
Results
Tools
View
Application
Find a Command
New FEM and Simulation
Context
Manage Materials
Physical Properties
Mesh Collector
Properties
More
Merge Face
Stitch Edge
Polygon Geometry
3D Tetrahedral
2D Mesh
Mesh Control
Mesh
Connections
Checks and ...
Show Only
Reverse Face Display
Show Adjacent
Show and Hide
Utilities
Menu
No Selection Filter
Entire Assembly
Simulation Navigator
Name
C.
fem3.fem
model3.prt
Polygon Geometry
2D Collectors
ThinShell(1)
Mesh Controls
CSYS
Groups
Fields
Modeling Objects (F...
Simulation File View
Preview
YC
XC
ZC
Y
X
Z

Step 3 : create material properties

a) On the **home page,** in the **properties** section, choose **More,** then **Assign Materials**

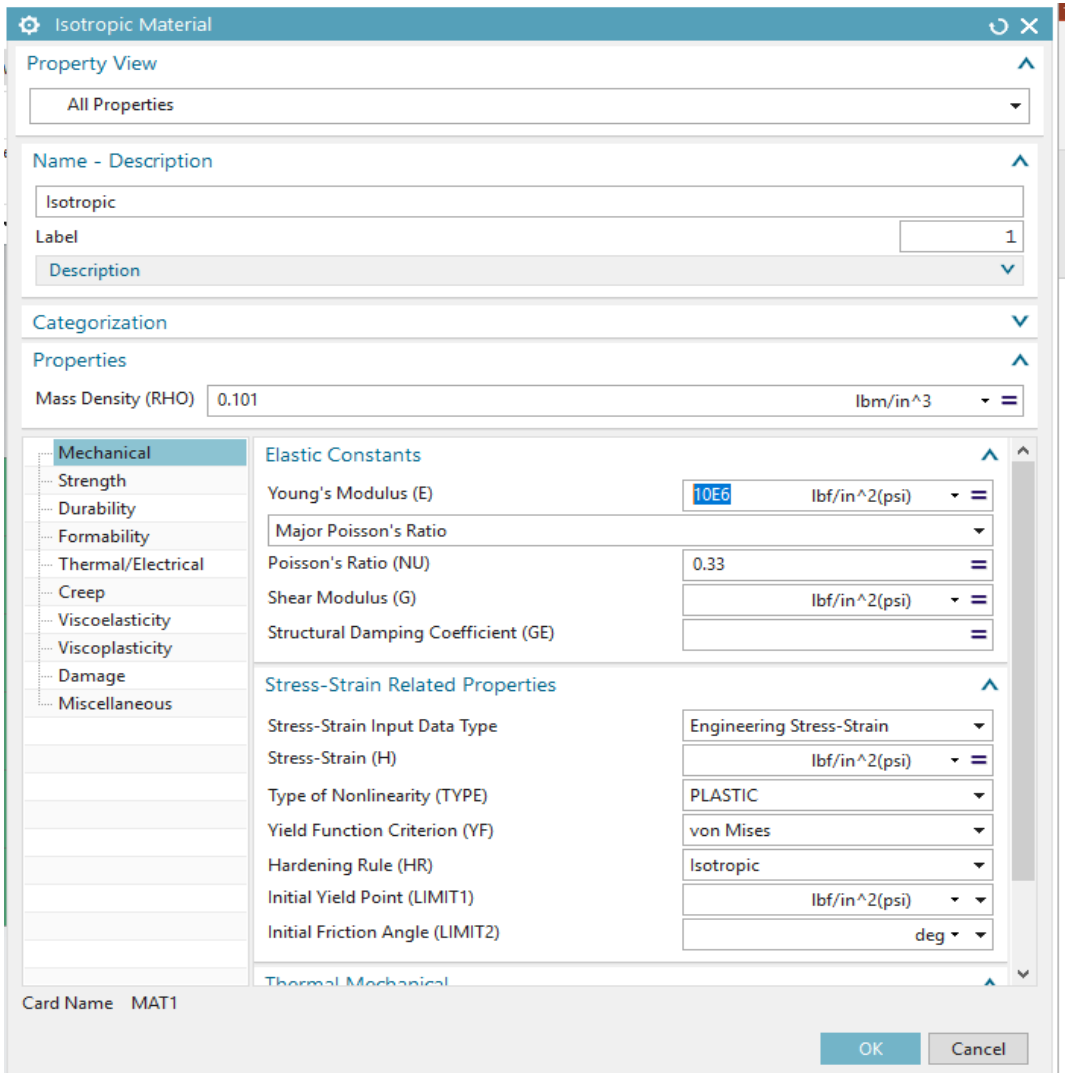
b) Select the rectangle

c) Then **Create Material** using the button on the right corner



a) Put the specified material properties as it is shown on the screen

b) Click **Ok**



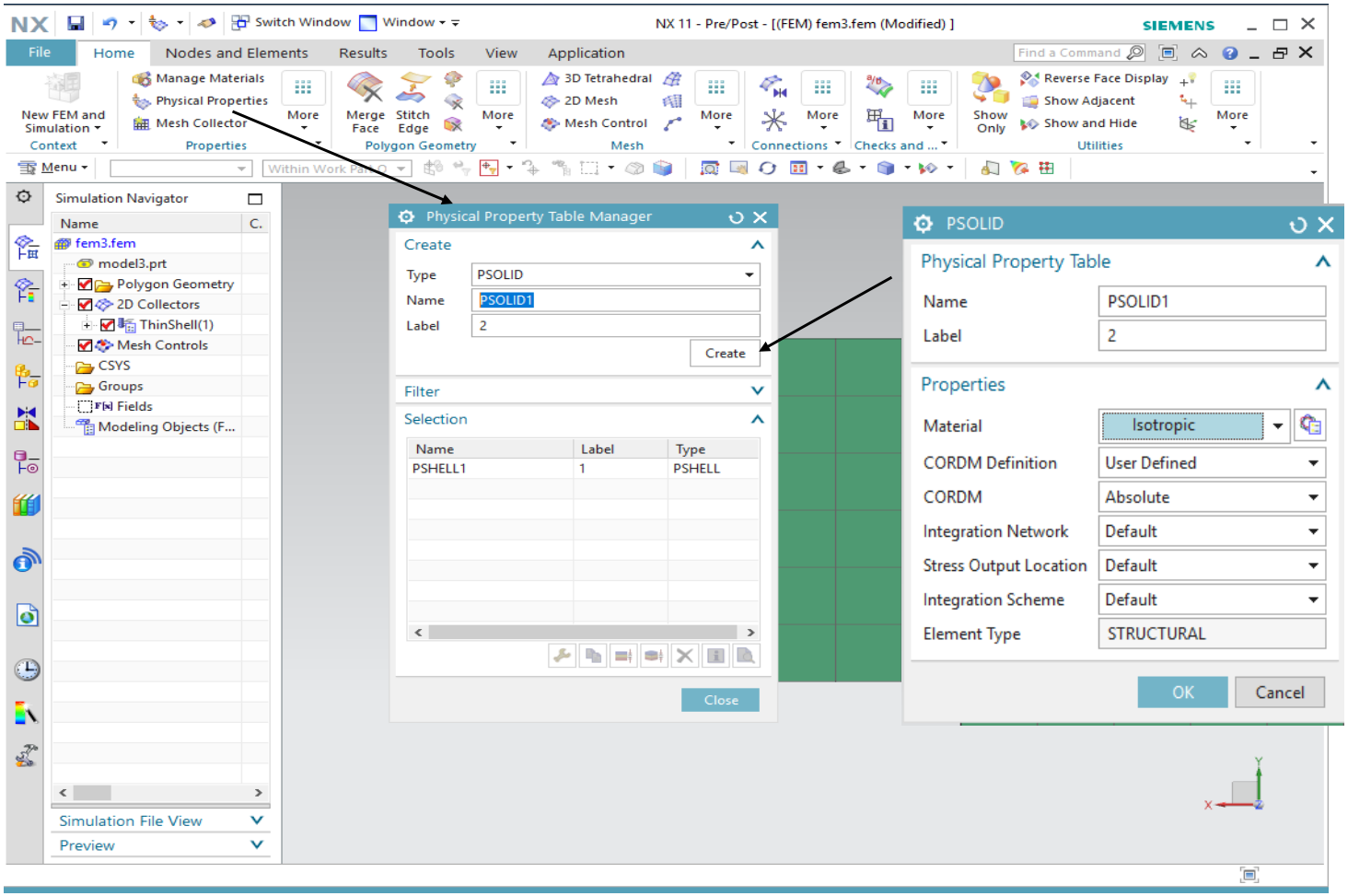
Step 4 : the physical properties

a) Choose **Physical Properties** and **Create**

b) Set the **Material** to be Isotropic

c) Click **Ok**

d) Close

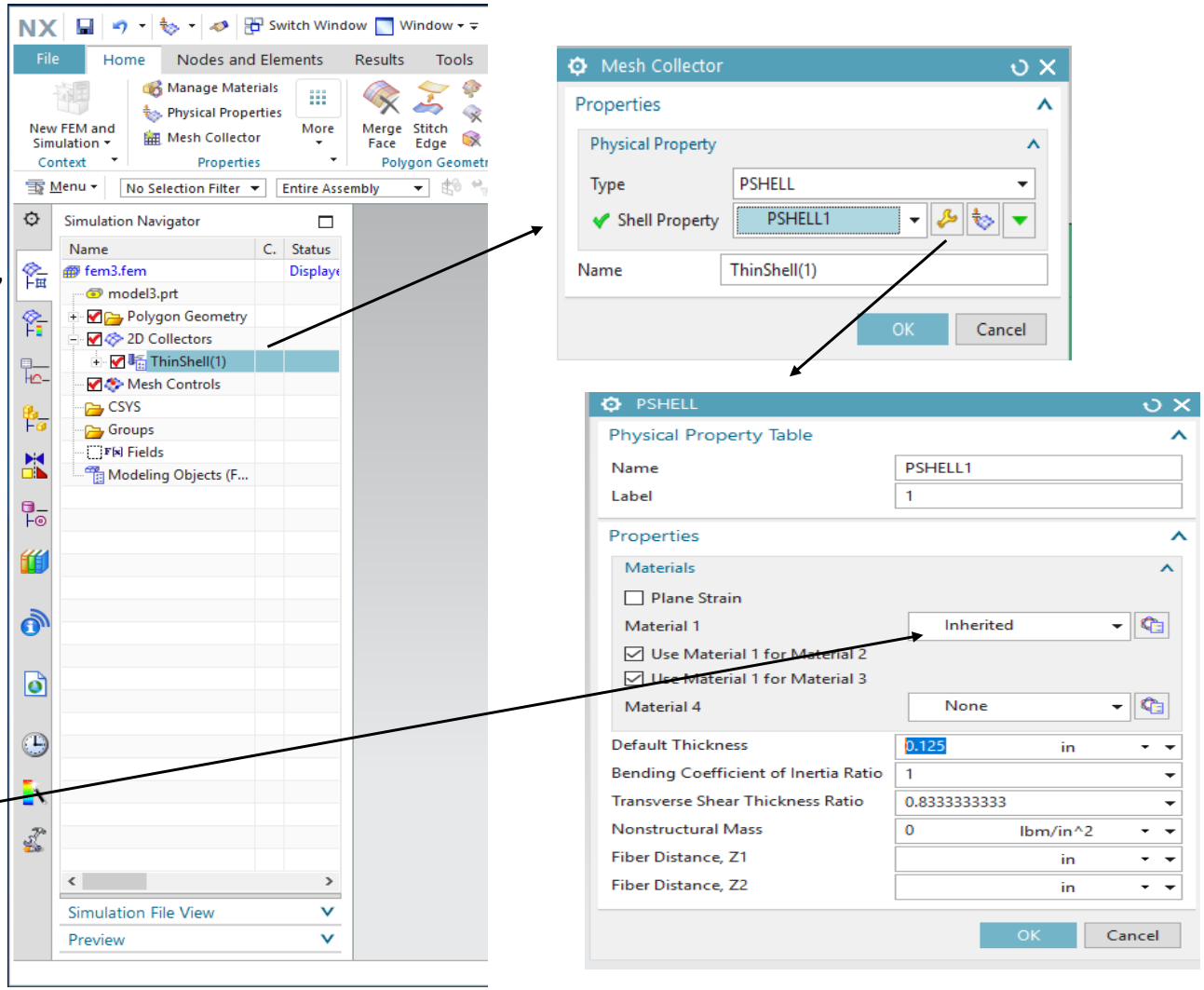


a) In the **Simulation Navigator** bar, extend the **2D Collectors**, then right click on **Thin shell,** choose **Edit**

b) Click on Edit (Shell Property), insert the **default Thickness: 0.125 in**

c) Material 1 should be set to **Isotropic**

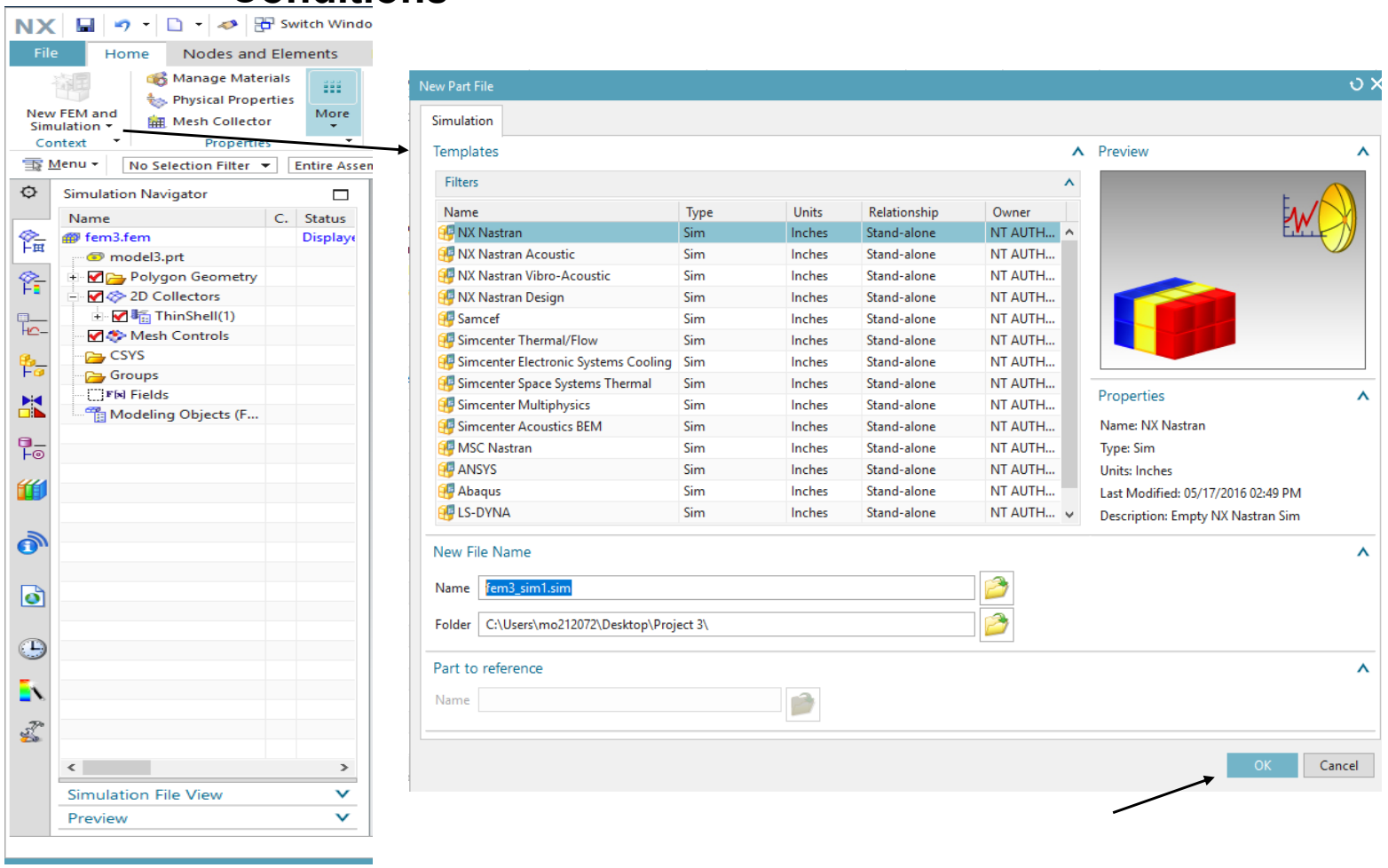
d) Click **Ok / Ok**



Step 5 : Apply Boundary Conditions

a) On the top left corner, click on **New FEM and Simulation,** then choose **New Simulation**

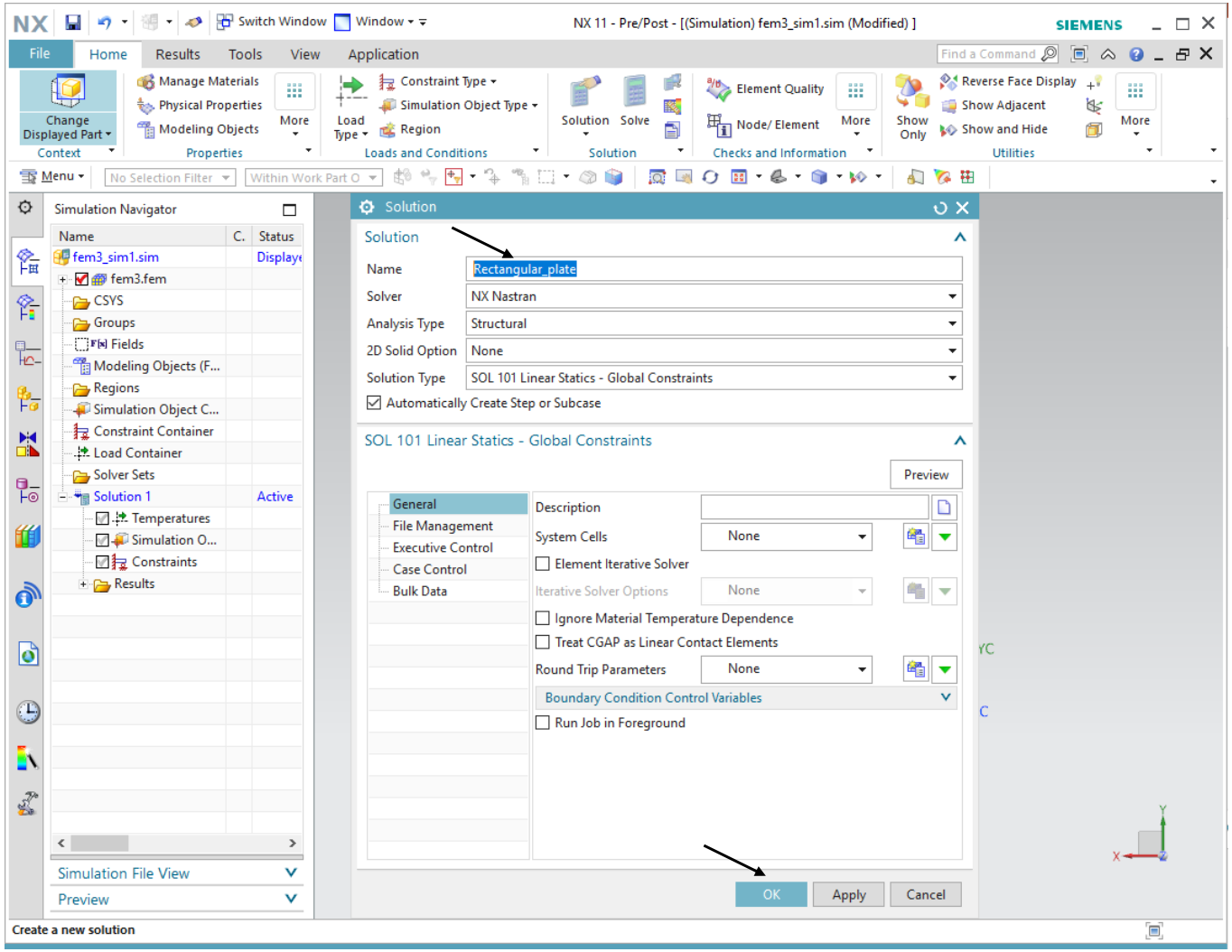
b) Click **Ok / Ok**



a) Name the solution

“**rectangular_plate**”

b) Click **Ok**

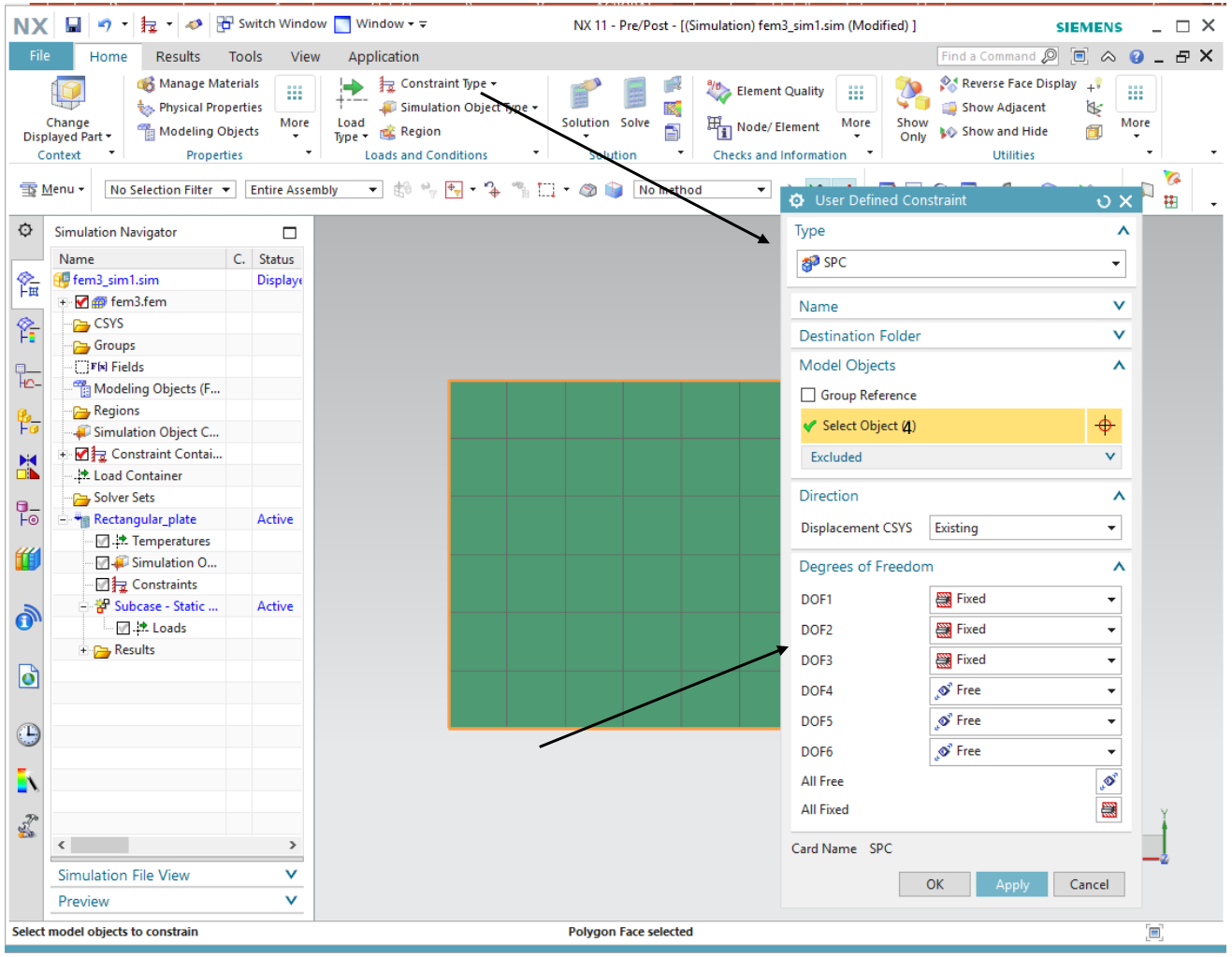


a) Click on **Constraint Type/ User Defined Constraint**

b) Then, **select all four edges**

c) Change the **Degrees of Freedom** as it is shown on the screen

d) Click **Ok**



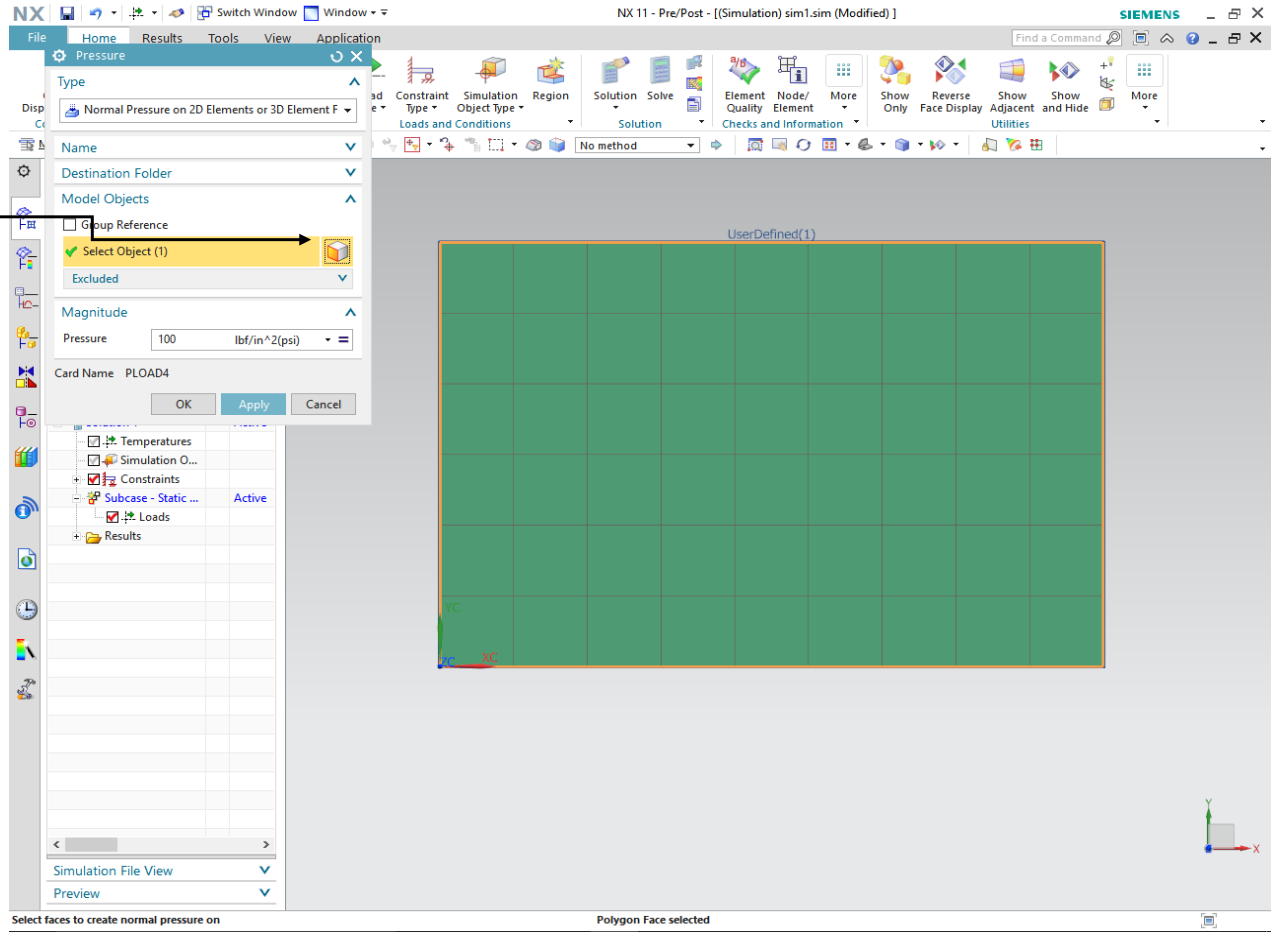
Step 5 : Apply Loads

a) Click on **Load Type/Pressure**

b) Then, activate the **Faces to Create Normal Pressure On** icon (box indicated here) and **select the rectangle**

c) Set pressure to **100 psi**

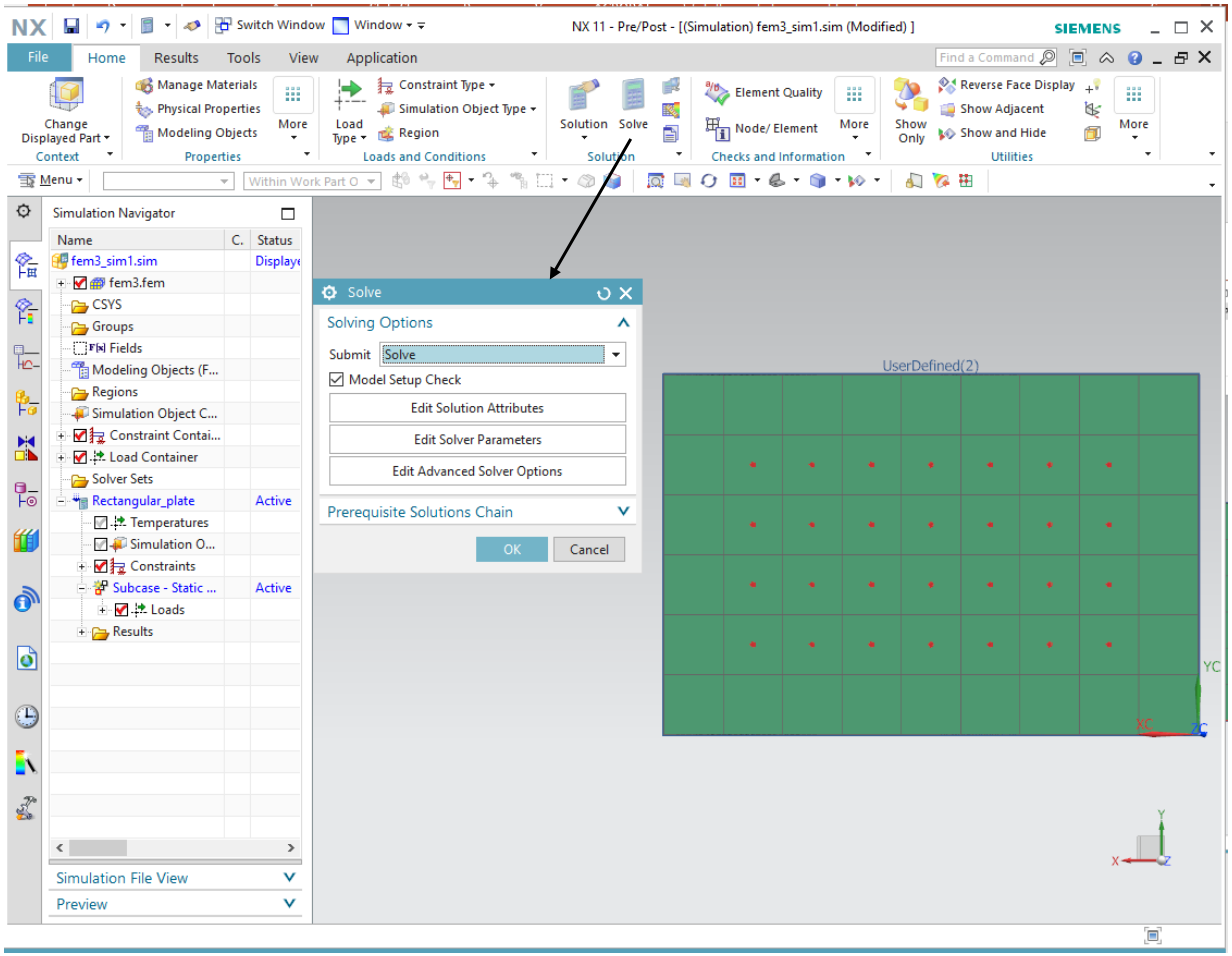
d) Click **Ok**



Step 5 : Run

a) Click on **Solve** in the home page

b) Click **Ok**

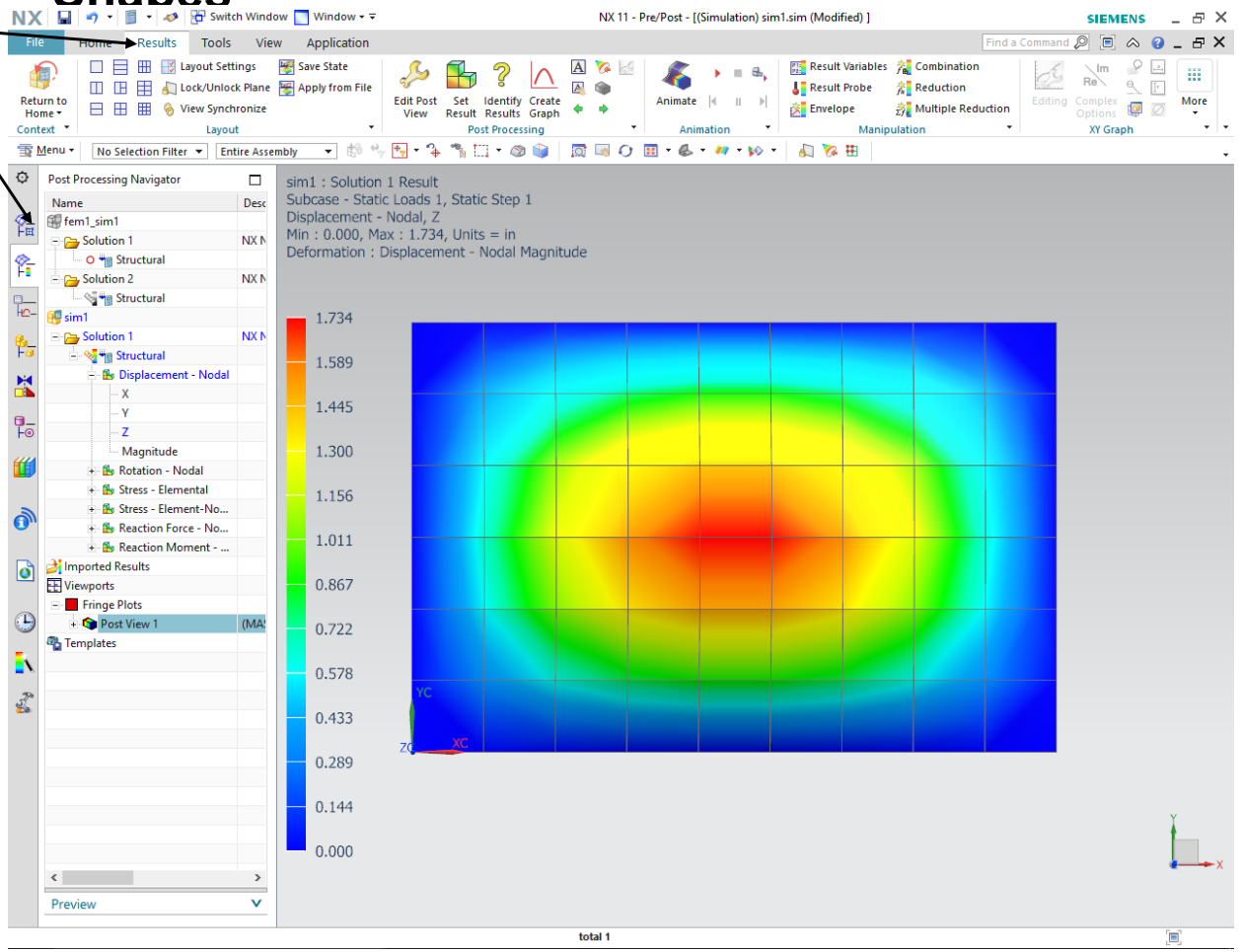


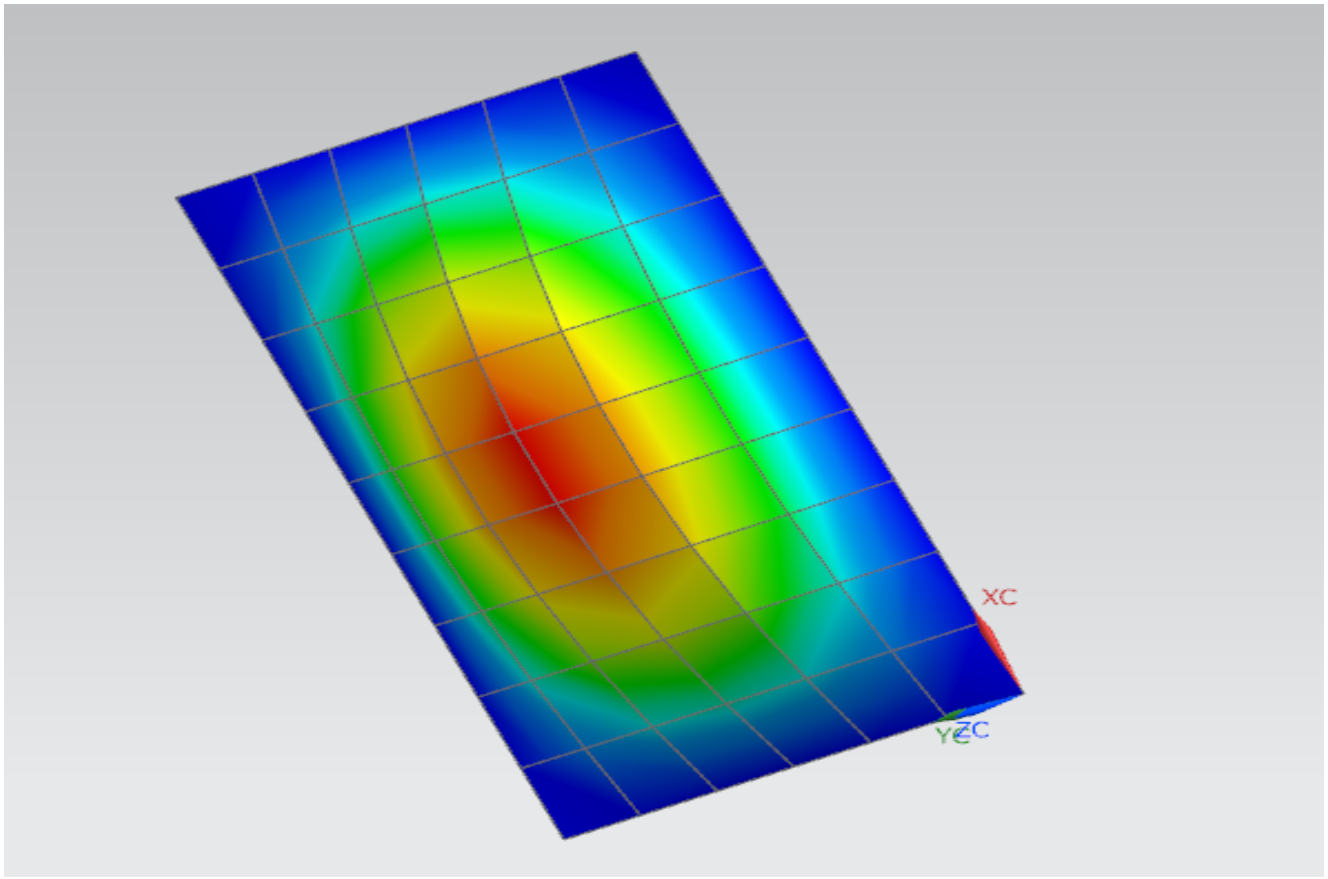
Step 5 :Plot the deformed Shapes

a) Open the **Results** section

b) In the **Simulation Navigator** bar, extend **Results** and double-click on **Structural**

c) Once the **Post Processing Navigator** bar pops up, extend **Structural/ Displacement – Nodal** and double-click on **Z**




XC
YC
ZC