1. Rectangular Plate under Uniform Tension (Plane Stress)

http://www.youtube.com/watch?v=1prWpg7vhQY

New features: Plane Element

i. General Definitions

For Plane problems, usually we work in the **XY Plane** instead of the XZ used for Frame structures.

For simplicity, the grid system should look like this one

---

3 Grids for X and Y centered at 0. Z is not used.

---

Units kN, cm, C
Mat. Steel
Self-Weight = 0
Thickness = 3mm
ii. Material and Section Definition
Define a Material having $E = 200 \text{ GPa}$ and $v = 0.3$.
In section definition, choose Menu Define > Section Properties > Area Sections…

Select Plane in the section type, then click on Add New Section

iii. Drawing the Model
Draw the four plane elements in the XY plane using Draw Poly Area tool.
It is common to draw the quadrilateral element starting from the lower left corner and continue counter clockwise.

4, doubleclick
3
2
We need a finer mesh, to have more accurate results. Select the four elements and go to Menu Edit > Edit Areas > Divide Areas…

Some recommendations are given in the reference manual about the aspect ratio of the elements. One should follow them to get the most accurate results.

iv. Displacement Boundary Conditions
Even if there are no apparent supports in the initial problem, we must specify some restraint conditions for the program to avoid a singular stiffness matrix. In this case, due to geometric and loading symmetry, we know that the central node will not move in either X nor Y direction. Also, all nodes on the Y axis will not move in the X direction.

Select all nodes on the Y axis and specify Translation Restraints in direction 1, then select the node at the origin and specify Translation Restraints in both Directions, 1 and 2.

v. Loading Condition
Two possible ways exist for applying the loads; by loading the nodes or by loading the elements.

Nodes (joints)
For the right side, the total load is \( F = (2 \text{kN/cm}^2) \times (4 \text{ cm}) \times (0.3 \text{ cm}) = 2.4 \text{kN} \) in the +X direction. We also have 5 nodes, 3 medians and 2 corners. The load distribution will be as follow:
For median nodes \( f = 2.4 / 4 = 0.6 \text{kN} \). For corner nodes \( f = 0.6 / 2 = 0.3 \text{kN} \).
By symmetry, the same loads are applied to the left side.
**Element (Area)**

For the right side, we have a uniform pressure in the outward direction. Select the four right edge elements > Assign > Area Loads > Surface Pressure (All)…

For the left side, we only change face 2 by **face 4**.

**vi. Analyse the System**

In plane elasticity problems, only two displacements are considered. Menu Analyze > Set Analysis Options …

Then **Run** the analysis.

Remember to save your work, to neglect self-weight and to disable modal analysis.
vii. Display Output
We can see the deformed shape as:

And the internal stresses (S11) as:

viii. Discussing Results
As we may have expected we can say:
- In the displacement diagram, the effect of positive Poisson’s ratio is clearly seen.
- In the stress diagram, we have uniformly distributed longitudinal horizontal stresses over the whole solid. Neither shear stresses nor longitudinal vertical stresses exist in the solid.

2. Tension Connection plate http://www.youtube.com/watch?v=uvmdgD6aU0M

New features: Pipe and Plate Module

Only half of the structure will be modeled. As SAP2000 is not extremely powerful in 2D and 3D modeling, the structure will be drawn using three elemental parts.

Start a blank file with only grids. X(0,5,15,25), Y(-10,0,10), Z(0)

Set Auto-merge tolerance to 0.1 mm, Menu Option > Dimensions/Tolerances…
- Start with rounded plate with circular hole

Menu Edit > Add Model From Template … then click on **Pipes and Plates**

![Model Selection Dialogue Box](image)

The main parameters are highlighted.

![Model With Parameters](image)

We need to delete the left half part of the circle, and reduce the number of nodes. For this purpose it is better to leave only one slice of the model, merge all the elements then divide them into 4 elements in the vertical direction. The final step is replicating the slice to reproduce the final form.
Select all elements in the previous screen and apply Replicate command, from the Edit Menu. Make sure to enter the parameters as in the figure below.
Save your file from time to time!
- Rectangular plate with circular hole

Menu Edit > Add Model From Template … then click on **Pipes and Plates**

We should remove the right half side of the rectangular plate and move the remaining part 15mm in the X direction (right), using the Move command from the Edit Menu.
- Last Rectangular part
We draw a big rectangle and divide it into 4 parts in X direction and 16 parts in Y Direction.

The final model should look like the figure above.

Due to Symmetry, the same boundary conditions as in the first example will be applied here as well. One concentrated nodal load will be applied to the node at coordinate (20, 0) \( F = 0.1\text{kN} \) in X direction.

For the analysis we will only consider displacements in the X and Y directions.

Run the Analysis
Output

Time for some nice pictures

Deformed shape (effect of concentrated Load)  Resultant Displacement Contour Lines

The three stress components in the local axis and the Von Mises stress.

Another efficient way to model two dimensional and three dimensional structural problems is to use the import functionality of SAP2000. Many formats are supported by the program, but the data must be prepared in a strict form. The same Problem 2 will be done using .DXF import function.

It is possible to use a simple modeling software such as Automesh2D, freely available from the website [http://www.automesh2d.com/default.htm](http://www.automesh2d.com/default.htm).

The geometric file used by this software is a text file containing the following data
The next step is to mesh the structure using the easiest auto-mesh function. 250 elements are chosen.

The final step in this software is to export the meshed structure in dxf format.

Before importing the model into SAP2000, it is usually recommended to edit the mesh in Autocad and copy all lines in a specific layer. This layer will be used by SAP2000 to import the appropriate model.

In SAP2000, use Menu File > Import > Autocad .dxf File … the select your file.
The next step is to draw a Plane element enclosing the whole imported model. Remember we imported the model as frame elements, but we need Plane element.

The final step is to divide the big plate element into the shape of the little frame lines. Select all and Menu Edit > Edit Areas > Divide Areas …

Delete the unwanted plane parts, the central circle and the two big corners. Delete also the frame elements as they don’t belong to our desired model.

Done! At least for the geometry modeling part. See video for more details.
4. Additional Examples

- **Plane Stress**
  Repeat Problem 1, add a uniformly distributed load $= 1 \text{kN/cm}^2$ in compression at the top and bottom faces of the plate.

- **Cantilever Beam (Thick theory)**

  Follow the same steps as Problem 1, take care about unit conversion. The load is uniformly distributed over the length, must be converted to node concentrated forces or uniformly distributed pressure.

  - Compare this solution with your classic beam analysis. Check deflection and maximum stress.

- **Bracket**
  Compare the solution of the given example Bracket.SDB with the one given by Ansys. The web link is a pointer to the University of Alberta tutorial for Ansys.

  [http://www.mece.ualberta.ca/tutorials/ansys/BT/Bracket/Bracket.html](http://www.mece.ualberta.ca/tutorials/ansys/BT/Bracket/Bracket.html)

The Model under SAP2000  
3D model from the U of A website