# Abaqus/CAE tutorial – 2D plane stress

Prepared by Dr. Hamed

## **Problem description**



The steel bracket shown is fixed to a wall along its left side. A tensile force with a magnitude of 5000N is applied to the right side of bracket. Determine the displacement and stress field in the bracket.

Note that:

- Small thickness of the bracket and in-plane loading motivate the use of 2D plane stress analysis.
- The structure contains a line of **symmetry**. So, only half of it will be modeled.

- (1) Start Abaqus and choose to create a new model.
- (2) In the model tree, double click on "parts" node
- (3) In the Create Part dialog box, name the part and select:
  - 2D planar
  - Deformable
  - Shell
  - Approximate size=2

| Model Results  |                                 |
|--|---------------------------------|
| 🥞 Model Database 🛛 🝷 🖹 🗞 🍟   |                                 |
|  | Create Part X                   |
| ⊟ <u>Model-1</u>   | Name: Bracket                   |
| Part 2   | - Modeling Enace                |
| Parts  | Modeling space                  |
| Sections   | C 3D 📀 2D Planar C Axisymmetric |
| in ∰r Profiles   |                                 |
| 비 455embly   | Type Options                    |
| Eield Output Requests  | Deformable                      |
| History Output Reguests  |                                 |
| Time Points  | C Discrete rigid None available |
| 📲 🛄 ALE Adaptive Mesh Constraints  | C Analytical rigid              |
| Interactions   |                                 |
| - 🔁 Interaction Properties   | Base Feature                    |
| Contact Controls   | G shall                         |
| Constraints  | o Shell                         |
|  | C Wire                          |
|  | C Point                         |
|  |                                 |
| - L BCs  |                                 |
| - 🦶 Predefined Fields  |                                 |
| Remeshing Rules  |                                 |
| L Sketches   | Approximate size: 2             |
| Annotations  |                                 |
| □ ♣ Analysis   | Continue Cancel                 |
| Bar Adapti itu Dupana  |                                 |
| The second secon |                                 |

#### (4) Create the geometry shown below.



(4) Double click on the "Materials" node in the model tree

- Name the material and give it a description
- Click on the "Mechanical" tab  $\rightarrow$  Elasticity  $\rightarrow$  Elastic
- Define Young's modulus and Poisson's ratio in SI units

📥 Edit Ma

|  | W   |
|--|---|
| Edit Material  | Name: Steel   |
| Name: Steel Description: Linear Isotropic Steel (SI units) Material Behaviors  | Description:<br>Material Behaviors<br>Elastic   |
| General       Mechanical       Ihermal       Other         Elasticity       Elastic         Plasticity       Hyperelask         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Hypegelastic         Damage for Fiber-Reinforced Composites       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Hypelestic | <u>General Mechanical Ihermal Other</u><br>Elastic<br>Type: Isotropic   |
| and a second   | Number of field variables: 0   Moduli time scale (for viscoelasticity): Long-term   No compression   No tension   Data   Young's   Poisson's   Modulus   Ratio   1   210e9   0.29 |

#### (6) Double click on the "Sections" node in the model tree

- Name the section "PlaneStressProperties" and select Solid for the category and "Homogeneous" for type
- Select the material created above (Steel) and set the thickness to be 0.01

| - Create Se            | ction                    | - Edit Section                      |
|------------------------|--------------------------|-------------------------------------|
| Name: plan<br>Category | ne_stress_tutoria        | Name: pl_stress_tutorial            |
| Solid                  | Homogeneous              | Type. Solid, Homogeneous            |
| Shell                  | Generalized plane strain | Material: Steel 👻 💆                 |
| 🔘 Beam                 | Composite                |                                     |
| Fluid                  |                          | Plane stress/strain thickness: 0.01 |
| Other Continue         | ue Cancel                | OK Cancel                           |

(7) Click on the "Assign Section" icon

- Select the surface geometry in the viewport
- Make sure that the section created above (plane\_stress\_tutorial) is selected.

| - Edit Se    | ction Assignment                                 |                               |
|--------------|--|-------------------------------|
| Region       |  |                               |
| Region:      | (Picked)   |                               |
| Section      |  |                               |
| Section:     | pl_stress_tutorial                               | ▼ <sup>1</sup> / <sub>2</sub> |
| Note: L<br>a | ist contains only sect<br>pplicable to the selec | tions<br>ted regions.         |
| Type:        | Solid, Homogeneo                                 | us                            |
| Material:    | Steel  |                               |
| Thickne      | ss   |                               |
| Assignm      | ent: 🔘 From sectio                               | n 🔘 From geometry             |
|              | ОК   | Cancel                        |



- (8) Expand the "Assembly" node in the tree model and then double click on "Instances"
  - Select "Dependent" for the instance type

|                                | 🔤 Create Instance 🛛 🔀                  |
|--------------------------------|--|
| Model Results                  | Parts                                  |
| 🥌 Model Database 🔄 韋 🛍 🇞 🍄     | Bracket                                |
| 🔁 🏭 Models (1)                 |  |
| E Model-1                      |  |
| 🕀 🕒 Parts (1)                  |  |
| 🕀 🎦 Materials (1)              |  |
| 🕀 🧱 Sections (1)               | <b> </b>                               |
| Profiles                       |  |
| 🛱 🏭 Assembly                   | Instance Type                          |
|                                | Opendent (mesh on part)                |
| - MP Position - Sockapists<br> | C Independent (mesh on instance)       |
| - 👉 Sets                       | Note: To change a Dependent instance's |
| Coppeting Ansions              | mesh, you must edit its part's mesh.   |
| A CALLER BERK                  | Auto-offset from other instances       |
|                                | OK Apply Cancel                        |

- (9) In the model tree, under the expanded "Assembly" node, double click on "Sets"
  - Name the set "Fixed"
  - Select the left edge of the surface in the viewport

| Model Results   |                                       |                       | _      |
|---|---------------------------------------|-----------------------|--------|
| 🧧 Model Database 🛛 👤 🌲 😵  | 📑 Create :                            | Set 👂                 | <<br>I |
| <ul> <li>Models (1)</li> <li>Model-1</li> <li>Parts (1)</li> <li>Materials (1)</li> <li>Sections (1)</li> <li>Sections (1)</li> <li>Profiles</li> <li>Assembly</li> <li>Instances (1)</li> <li>Me Position Constraints</li> <li>Features (1)</li> </ul> | Name: Fixe<br>Type: Geom<br>Continue. | ed<br>netry<br>Cancel |        |
| - 👉 Sets<br>- 🦢 Surfaces<br>- 📴 Conr Sets Assignments   |                                       |                       |        |

## (9) (continue)

- Create another Set named "Symmetry"
- Select the two lower horizontal edges of the surface in the viewport
- (10) In the model tree, under the expanded "Assembly" node, double click on "Surfaces"
  - Name the surface "PressureLoad"
  - Select the right edge of the surface in the viewport

| Model Results  |                    |   |
|--|--------------------|---|
| 🥞 Model Database 🔄 韋 🛍 🇞 🍟   |                    |   |
| 🖻 🏭 Models (1)   | Create Surface     | × |
| ■ Model-1<br>⊕ Parts (1)<br>⊕  | Name: PressureLoad |   |
|  | Type: Geometry     |   |
| <ul> <li>Assembly</li> <li>Instances (1)</li> <li>IP Position Constraints</li> </ul> | Continue Cancel    |   |
| <ul> <li></li></ul>  |                    |   |
| - 🔁 Connect è Accienments  |                    |   |

(11) Double click on the "Steps" node in the model tree

- Name the step, set the Procedure to "General" and select "Static, General".
- Give the step a description

| Model Results   | Create Step  | × | Edit Step   |
|---|--|---|---|
| Se Model Database   | Name: Apply Load   |   | Name: Apply Load<br>Type: Static, General   |
|   | Insert new step after  |   | Basic Incrementation Other  |
| Model-1  Mo | Initial  |   | Description: Apply pressure load to bracket Time period: 1 NIgeom: Off (This setting controls the inclusion of nonlinear effects O on of large displacements and affects subsequent steps.) Automatic stabilization: None |
|   | Procedure type: General Dynamic, Explicit Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General Static, Riks Continue Cancel |   |   |

(12) Expand the Field Output Requests node in the model tree, and double click on F-Output-1

- Uncheck the variables "Strains" and "Contact"
- Give the step a description

| Model Results                   | Edit Field Output Request   |
|---------------------------------|---|
|                                 | Name: F-Output-1  |
| 😂 Model Database 🛛 💆 葦 🛅 🎭 🏆    | Step: Apply Load  |
|                                 | Procedure: Static, General  |
|                                 | Domain: Whole model   |
|                                 | Frequency: Every n increments n: 1  |
|                                 | Timing: Output at exact times   |
| H / C Materials (1)             |   |
| 🕀 🔆 Sections (1)                | Output Variables  |
| Profiles                        | Select from list below O Preselected defaults O All O Edit variables                |
| 🕀 🏙 Assembly                    | S,U,RF,CF   |
| 🗄 야🛱 Steps (2)                  | ► 🔽 Stresses  |
| 🖻 🏪 Field Output Requests (1)   | ▶   |
| F-Output-1                      | Implacement/Velocity/Acceleration   |
| 🕀 📴 History 🖾 Eput Requests (1) | Forces/Reactions  |
|                                 | Contact   |
|                                 | F Energy  |
|                                 | Failure/Fracture  |
|                                 |   |
|                                 |   |
|                                 | Note: Error indicators are not available when Domain is Whole Model or Interaction. |
|                                 | Output for rebar  |
|                                 | Output at shell, beam, and layered section points:                                  |
|                                 | Use defaults C Specify:   |
|                                 | ☑ Include local coordinate directions when available                                |
|                                 | OK  |

- (13) Expand the History Output Requests node in the model tree, right click on H-Output-1 and select Delete
- (14) Double click on the "BCs" node in the model tree
  - Name the BC "Fixed" and select "Displacement/Rotation" for type
  - In the prompt area click on the Set button
  - Select the set named "Fixed"
  - Check U1 and U2 to fully restrain the left edge

| Fields     Amplitudes     Loads     Prevent Fields     Renearing Rules     Sketches     Annotations     Sketches     Analysis     Jobs     Adaptivity Processes  | <ul> <li>Greate Boundary Condition</li> <li>Name: Fixed</li> <li>Step: Step-1</li> <li>Procedure: Static, General</li> <li>Category</li> <li>Mechanical</li> <li>Fluid</li> <li>Other</li> <li>Other</li> <li>Symmetry/Antisymmetry/Encastre</li> <li>Displacement/Rotation</li> <li>Velocity/Angular velocity</li> <li>Connector displacement</li> <li>Connector velocity</li> </ul>  |
|--|--|
| Region Selection         Eligible Sets         Sets below may contain vertices, edges, faces, cells or nodes.         Name filter:         Image: Sets below may contain vertices, edges, faces, cells or nodes.         Name filter:         Image: Sets below may contain vertices, edges, faces, cells or nodes.         Name filter:         Image: Sets below may contain vertices, edges, faces, cells or nodes.         Name filter:         Image: Sets below may contain vertices, edges, faces, cells or nodes.         Name       Type         Fixed       Geometry         symmetry       Geometry         Image: Sets below may contain viewport       Image: Sets below may contain viewport         Image: Continue       Dismiss | ►       It Boundary Condition         Name:       Fixed         Type:       Displacement/Rotation         Step:       Step-1 (Static, General)         Region:       (Picked)         CSVS:       (Global)         Distribution:       Uniform         Ull:       0         Ull: |

OK

Cancel

(14) (continue)

- Repeat the procedure for the symmetry BC using the set named "Symmetry". Set U2 to be 0 for this BC.
- (15) Double click on the "Loads" node in the model tree
  - Name the load "Pressure" and select "Pressure" as type
  - Select the surface named "Pressure"
  - The applied pressure is -5x10<sup>6</sup> N/m<sup>2</sup>, which is a load of -5000/2 N distributed across the right cross sectional area of surface (-2500 N/(0.05 m x 0.01 m))

| Connector Sections                         | Create Load                       | ×        |
|--|-----------------------------------|----------|
| ${old E}  {oldsymbol {\mathcal F}}$ Fields | Name: Pressure                    |          |
| - 🔄 Amplitudes                             | Step: Apply Load 💌                |          |
| - 🕒 Loads                                  | Procedure: Static, General        |          |
| 🗄 🛄 BCs 🖓                                  | Category Types for Selected 9     | Step —   |
| 🖳 🔛 Prede <mark> Loads</mark> elds         | Mechanical     Concentrated force |          |
| Remeshing Rules                            | C Thermal Moment                  |          |
| Sketches                                   | C Acoustic Pressure               |          |
|  | C Fluid Shell edge load           |          |
|  | C Electrical Pipe pressure        |          |
| - Region Selection                         | C Mass diffusion Body force       |          |
| Eligible Surfaces                          | C Other Line load                 |          |
| Surfaces below may contain faces.          | Gravity                           |          |
| Name filter:                               | Boic load                         | <b>_</b> |
| Name Type                                  |                                   | X        |
| Pressure Surface                           |                                   |          |
|  | Name: Load-1                      |          |
|  | Type: Pressure                    |          |
| V Highlight selections in viewport         | Step: Step-1 (Static, General)    |          |
| Continue Dismiss                           | Paging Descure                    |          |
|  | anegion: Pressure                 |          |
|  | Distribution: Uniform             | t(x)     |
|  | Magnitude: 5:6                    |          |
|  |                                   | in l     |
|  | Amplitude: (Ramp)                 | Ю        |
|  |                                   |          |

After application of BCs and load, your model should look like the figure below.



(16) In the model tree, double click on the "Mesh" for the Bracket part, and in the toolbox area click on the "Assign Element Type" icon

- Select "Standard" for element type
- Select linear for geometric order
- Select "Plane Stress" for family
- Pick the tab "Tri" and note the name and description of CST triangular element in the description window:
   CPS3

| Model       Results         Model Database       Image: Construct of the second s | Assign<br>Element Type | Flement Type          Element Library       Family         Image: Standard in Explicit       Plane Strain         Plane Stress       Pore Fluid/Stress         Dere Fluid/Stress       Thermal Electric         Quad Trip       Element Controls         Viscosity:       Image: Use default in Specify         Second-order accuracy:       Yes in No         Distortion control:       Image: Use default in Yes in No         Length ratio:       0.1 |
|---|------------------------|--|
|   |                        | Length ratio: 0.1<br>Element deletion: Outron: Use default Yes No  |
|   |                        | CPS3: A 3-node linear plane stress triangle.   |

(17) In the toolbox area, click on "Assign Mesh Controls"

Select the element shape as "Tri" and click OK

| L. <b>H-</b>  | 🐥 Mesh Controls               |                                      |  |  |  |
|---------------|-------------------------------|--------------------------------------|--|--|--|
|               | Element Shape                 |                                      |  |  |  |
|               | 🔘 Quad 🔘 Quad-dominated 💿 Tri |                                      |  |  |  |
| Assign        | Technique                     | Algorithm                            |  |  |  |
| Mesh Controls | As is                         | Use mapped meshing where appropriate |  |  |  |
|               | Free                          |                                      |  |  |  |
| 8. 🚍          | Structured                    |                                      |  |  |  |
|               | 🔘 Sweep                       |                                      |  |  |  |
|               | Multiple                      |                                      |  |  |  |
|               | ОК                            | Defaults Cancel                      |  |  |  |

(18) In the toolbox area, click on "Seed Part" icon

- Select the approximate global size to 0.02
- Pay attention to options for mesh quality control

|           | Sizing Controls   |
|-----------|---|
| Seed Part | Approximate global size: 0.02<br>Curvature control<br>Maximum deviation factor (0.0 < h/L < 1.0): 0.1<br>(Approximate number of elements per circle: 8) |
| à 💼       | Minimum size control<br>By fraction of global size (0.0 < min < 1.0) 0.1<br>By absolute value (0.0 < min < global size) 0.0035                          |
|           | OK Apply Defaults Cancel  |

Apply

(19) In the toolbox area, click on "Mesh Part" icon



(20) In the model tree double click on the "Job" mode

- Name the job "Bracket"
- Give the job a description

| III IIII IIII IIIIIIIIIIIIIIIIIIIIIIII |                 | Edit Job  | × |
|--|-----------------|---|---|
| Prederined Fields                      | Create Job X    | Name: Bracket                                       |   |
| Remeshing Rules                        | Name: Bracket   | Model: Model-1                                      |   |
|  | Source: Model   | Description: Static analysis of a bracket           |   |
| Annotations                            |                 | Submission General Memory Parallelization Precision |   |
| 🖻 🔹 🖡 Analysis                         | Model-1         | _ Job Type  |   |
| - 💻 Jobs                               |                 | <ul> <li>Full analysis</li> </ul>                   |   |
|  |                 | C Recover (Explicit)                                |   |
| Jobs Frocesses                         |                 | C Restart   |   |
| J                                      |                 | Run Mode  |   |
|  |                 | Background C Queue: Host name:     Type:            |   |
|  |                 | Submit Time   |   |
|  |                 | • Immediately                                       |   |
|  | Continue Cancel | O ∀ait: hrs min.                                    |   |
|  |                 | C At: Tip   |   |
|  |                 | OK  |   |

- (21) In the model right click on the job created (Bracket) and click on "Submit"
  - While Abaqus is solving the problem, right click on the submitted job (Bracket) and select "Monitor"
  - In the monitor window, check that there are no errors or warnings



(22) In the model tree right click on submitted and successfully completed job (Bracket) and select "Results"



- (23) Display the contour of Mises stress on the deformed shape
  - In the toolbox area click on the "Plot Contours on Deformed Shape" icon





(24) Request displacement plot along x direction, with the undeformed shape superimposed.



The image was obtained using the "Superimpose Options" icon and setting the parameters as shown. Note the clear stretching of the right side of the bracket and the imposition of symmetry BC that prevents lower horizontal surface from moving in the vertical direction.



| 🔶 Supe | rimpose Plot Op            | tions  |                |       | x    |  |  |  |  |
|--------|----------------------------|--------|----------------|-------|------|--|--|--|--|
| Basic  | Color & Style              | Labels | Normals        | Other |      |  |  |  |  |
| Rend   | Render Style Visible Edges |        |                |       |      |  |  |  |  |
| 🔘 Wi   | ireframe 🔘 Hid             | den 🔘  | All edges      |       |      |  |  |  |  |
| I Fil  | led 💿 Sha                  | ded 🔘  | Exterior edges |       |      |  |  |  |  |
|        |                            | C      | Feature ed     | ges   |      |  |  |  |  |
|        |                            | ۲      | Free edges     |       |      |  |  |  |  |
|        |                            | C      | No edges       |       |      |  |  |  |  |
|        |                            |        | ·`@`           |       |      |  |  |  |  |
|        |                            |        |                |       |      |  |  |  |  |
|        |                            |        |                |       |      |  |  |  |  |
|        |                            |        |                |       |      |  |  |  |  |
|        |                            |        |                |       |      |  |  |  |  |
| Ok     | К Арр                      | ly [   | Defaults       | Cance | el 🛛 |  |  |  |  |

(25) Activate the icon "results Options" on the toolbox area.

- Uncheck the box "Average element output at nodes"
- Change the stress output to S11.



 Note the stress value in each element is depicted only by a single color, denoting the constant stress nature of 3-node triangular element (CST)

