Introduction to StressCheck®

These slides complement the Getting Started Guide of StressCheck®. The step-by-step procedures of importing a Parasolid file, creating the mesh, entering input data, solving and post-processing are illustrated.

Manual construction of a mesh for a parameterized plane elastic body and specification of parameterized loading are illustrated on the basis of Exercise 4.3.9.

Introduction to StressCheck

- Import a Parasolid file
  - Start StressCheck and make sure that the Reference/Theory Toolbar is displayed.
    - It is good practice to display all toolbars: Click on View and select ALL Toolbars. We will not need the Part/Assembly Toolbar, turn it off.
  - Select *Planar / Elasticity / mm/N/sec/C*
    - The type of analysis and units must be consistent with the Parasolid file.
  - Select *File > Import*
    - A reminder will be displayed. If you did not update the units as indicated above, do so before importing the file.
    - Using the browser import your Parasolid file.
    - In this example we will import NotchedBeam1.x_t

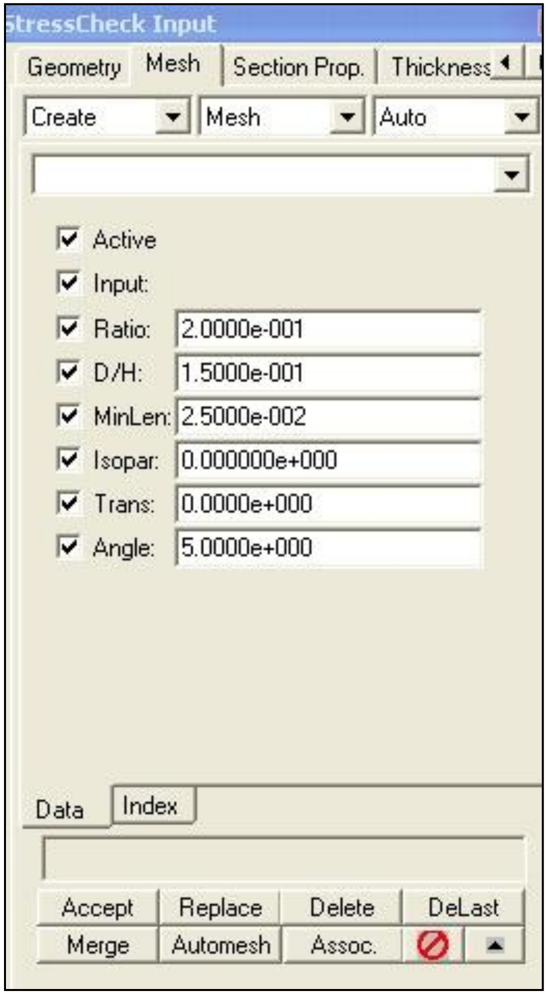
Create an automesh (1)

- You should see the Parasolid sheet body shown below.

- Select the icon  or click *Alt – I*. This will take you to the StressCheck Input dialog box.
- Click on the Mesh tab and set the scroll boxes at the top display to: Create/Mesh/Auto.

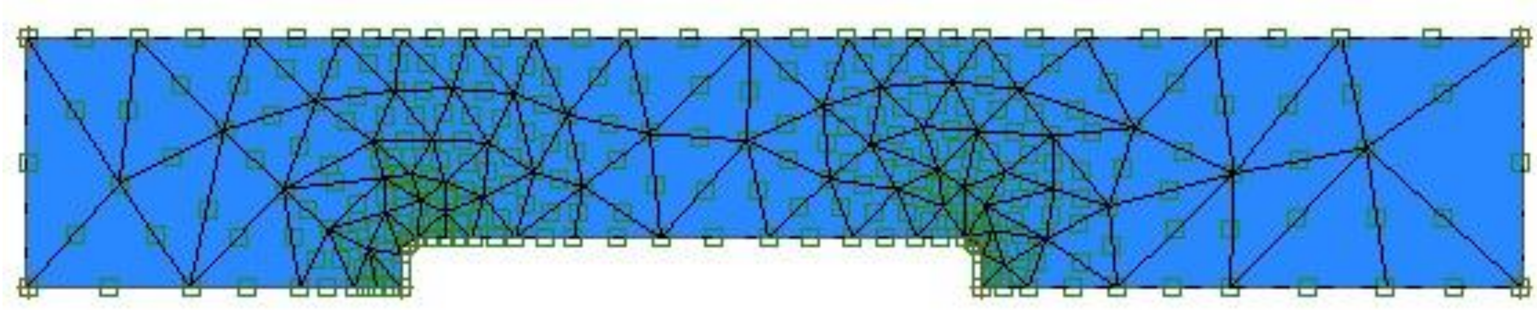
Create an automesh (2)

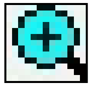
- You will see the dialog box shown on the right.
- Complete as shown and click on Accept. This means that you have accepted the default meshing parameters. A meshing record will be created.
- Click on Automesh. This will produce the mesh of 6-node isoparametric elements shown in next slide.

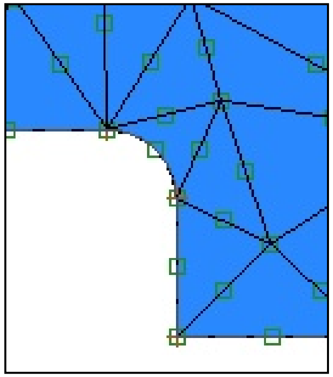


Create an automesh (3)

- The mesh shown below will appear



To zoom left click on the icon  and drag with the right button held.

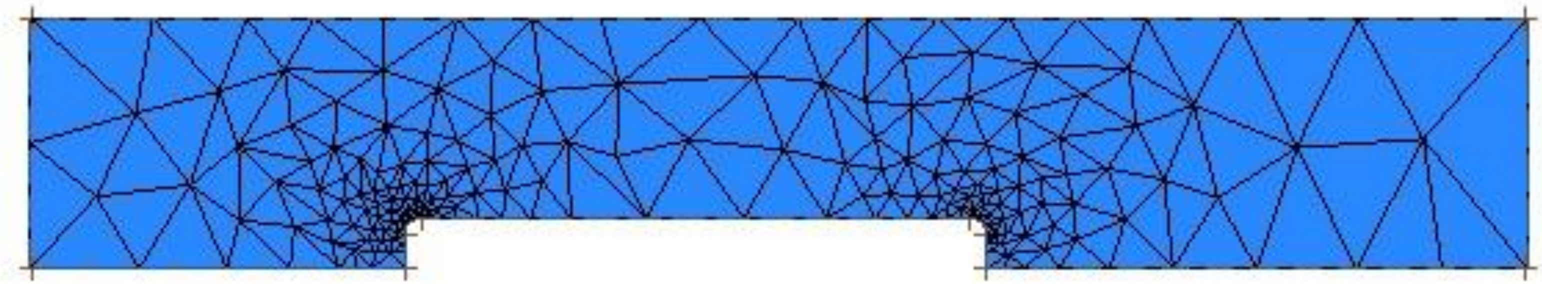


Zooming on the circular fillets you will see that the 90 – degree circular sector is approximated by only one element.

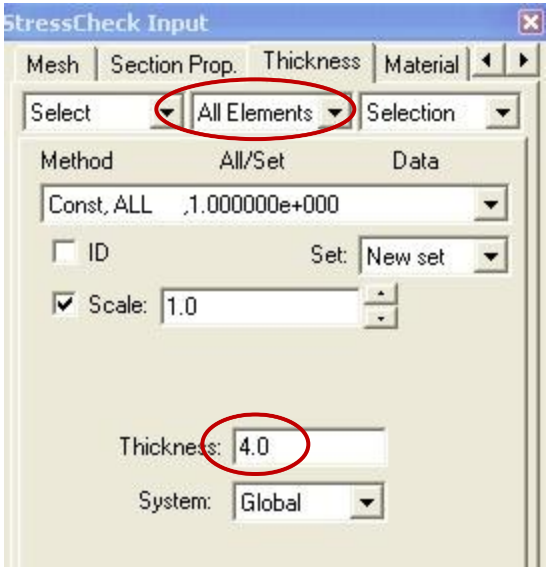
Smaller elements will be created at the fillet if you select Create / Mesh / Curve then select both fillets, use default values in the mesh interface, Accept > Automesh

Enter input records (1)

- The refined mesh is shown below



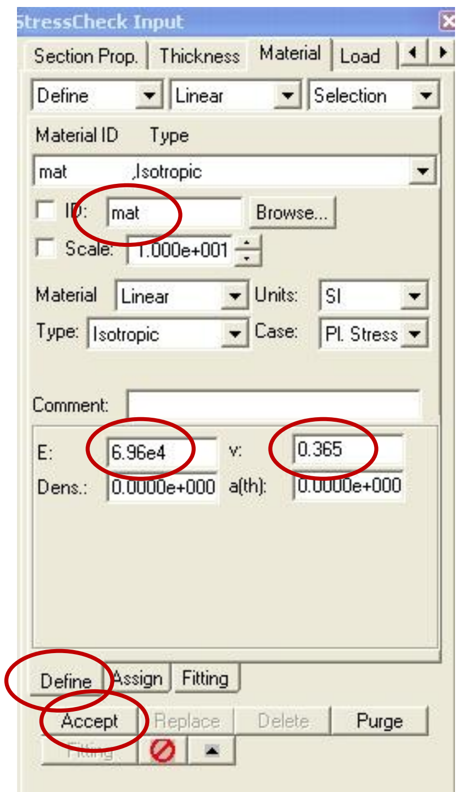
- Assign the thickness
  - *Select > All Elements*
  - Enter the thickness (4.0)
  - *Accept*



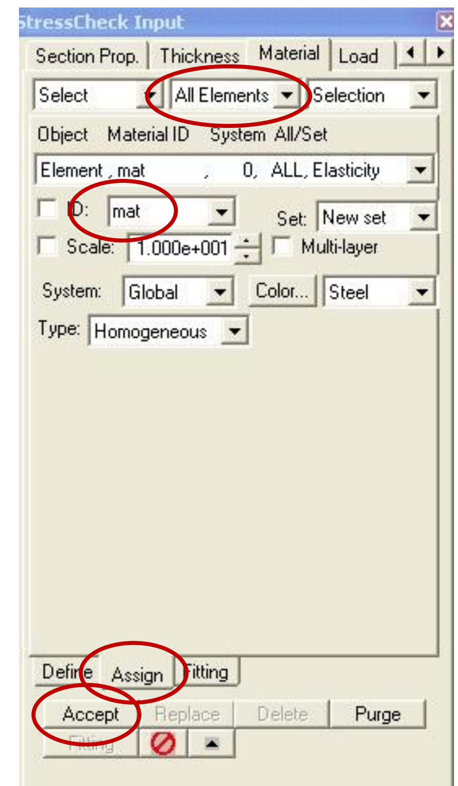
Enter input records (2)

- Material properties are entered in two steps.

Define

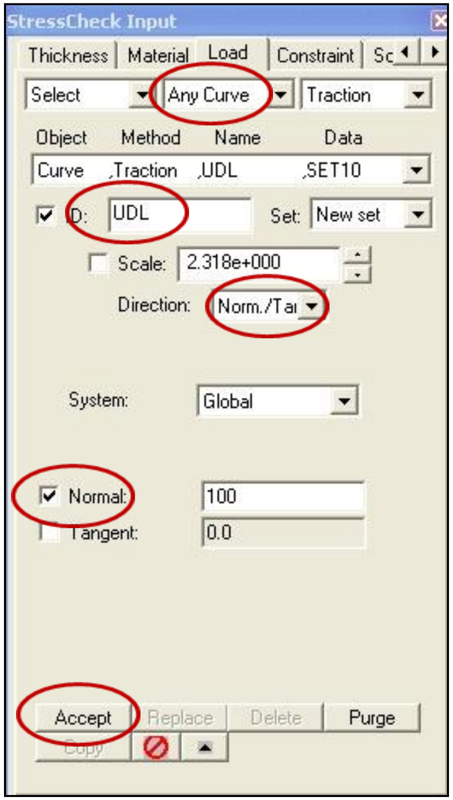


Assign



Enter input records (3)

- Load records
  - Let us enter uniformly distributed loads

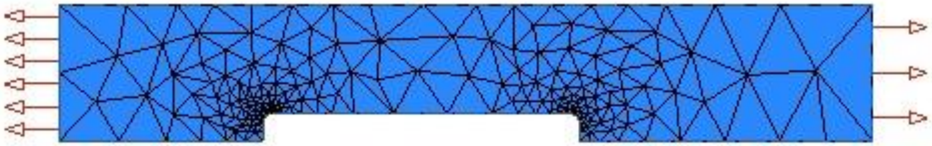


Select curve

Assign name of the load record

Specify direction

Specify component and value then Accept

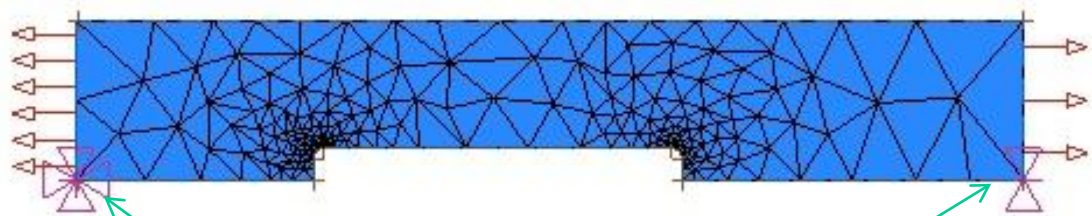


Enter input records (4)

- Constraint record

Let us enter a rigid body constraint record as shown in the figure on the left.

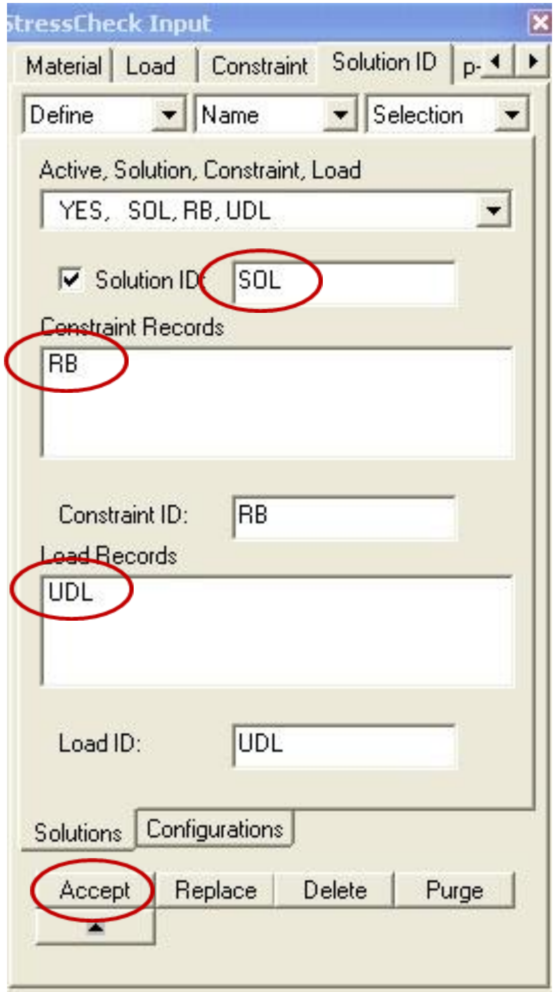
Select the two corner points as shown below and the rigid body constraint will be created automatically (no need to click on Accept).



Rigid body constraint

Enter input records (5)

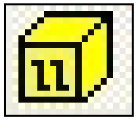
- Solution record
  - Associates the names of a constraint record (RB) and load record (UDL) with the name of a solution (SOL).
  - Click on Accept to create record.

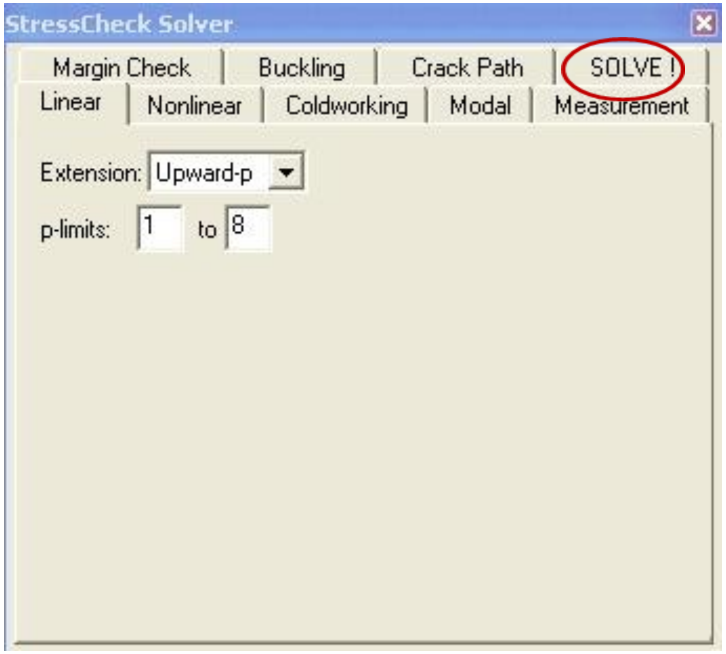


Recommended exercise

- In the preceding steps we first created the mesh then entered the input data.
- We could have entered the input data then created the mesh.
  - In general it is good practice to associate the load and constraint records with geometric objects (points, curves) rather than with meshing objects (nodes, element edges or edge curves).
- Enter the input data first then create the mesh.
  - The input data can be entered in any order.

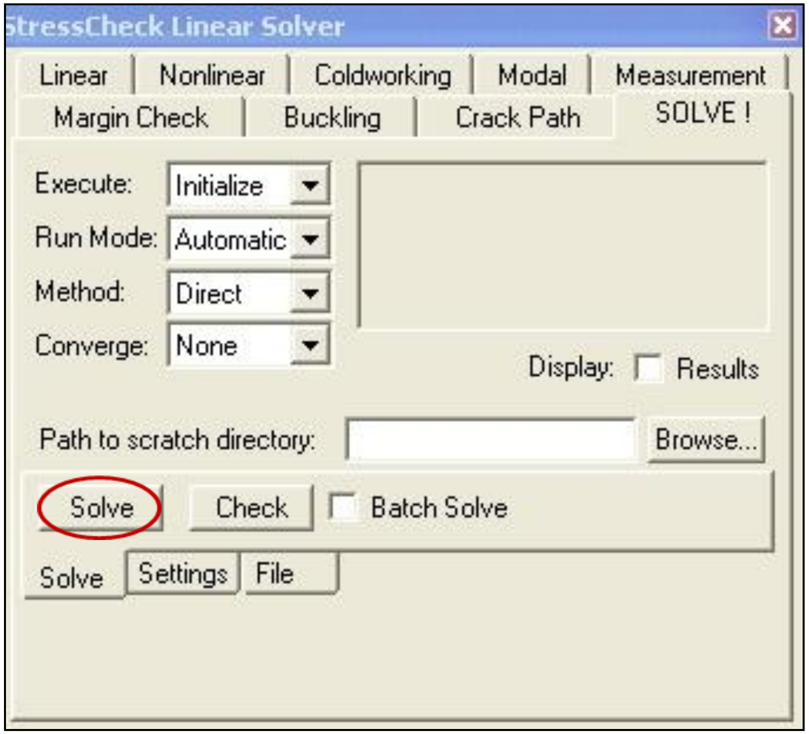
Solution (1)

- Click on the icon  or press Alt – S
  - You will see the solution tab below. Fill in the range of p values (in this case 1 to 8) and click on Solve.

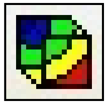


Solution (2)

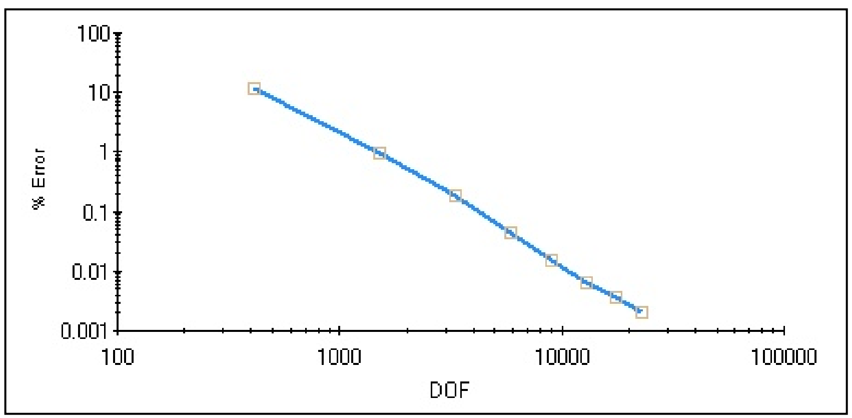
- You will see the second solution tab shown below.
  - Complete as shown and click on Solve.



Postprocessing (1)

- Click on the icon  or press Alt - X
- Select the Error tab, and complete the solution ID and run numbers, then Accept.
  - You will see a graph showing a the relative error in energy norm in the Chart tab, and you can select to see the tabular data in the Table tab.

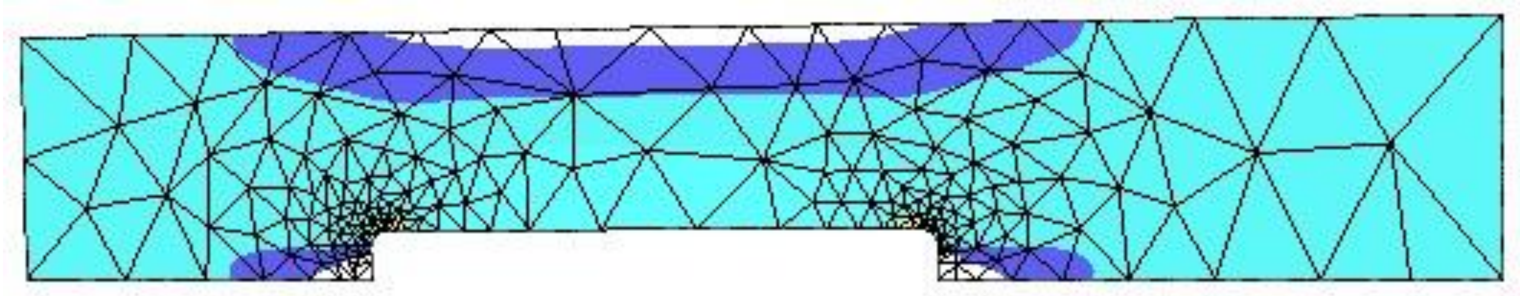
Relative error in energy norm



p	N	PE	beta	% err.
1	415	-1.199360E+06	0	11.72
2	1527	-1.215952E+06	1.93	0.95
3	3335	-1.216057E+06	2.12	0.18
4	5839	-1.216061E+06	2.51	0.04
5	9039	-1.216061E+06	2.5	0.01
6	12935	-1.216061E+06	2.25	0.01
7	17527	-1.216061E+06	2.08	0
8	22815	-1.216061E+06	2.08	0

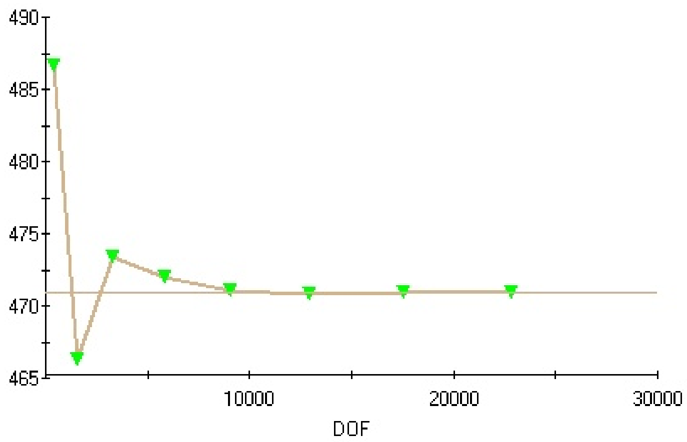
Postprocessing (2)

- Click on the Plot tab
  - Select the solution record and run number that you wish to plot. Here p=8 was selected.
  - For a contour plot click Fringe.
  - Select the function to be plotted. Here we selected Seq (the von Mises stress).
  - Specify the number of midsides. This determines the resolution of the data plot. Here 5 was selected.



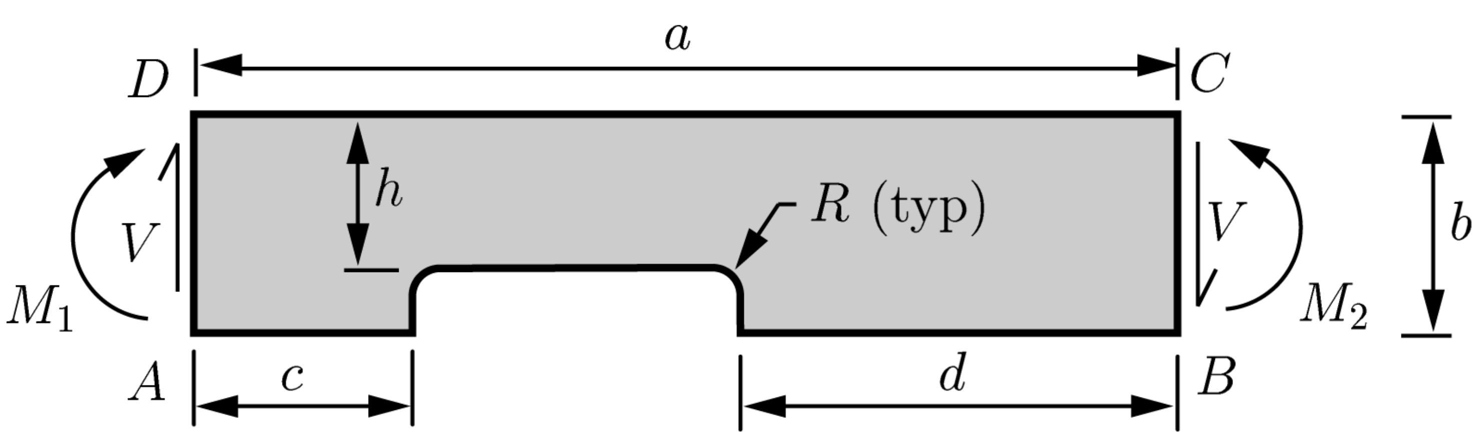
Postprocessing (3)

- Click on the Min/Max tab
  - Select the solution records p=1 to p=8.
  - Select the function of interest. Here we selected S1 (the first principal stress).
  - Specify the number of midsides. This determines the search grid. Here 9 was selected. Click on Accept and you will see the graph below.



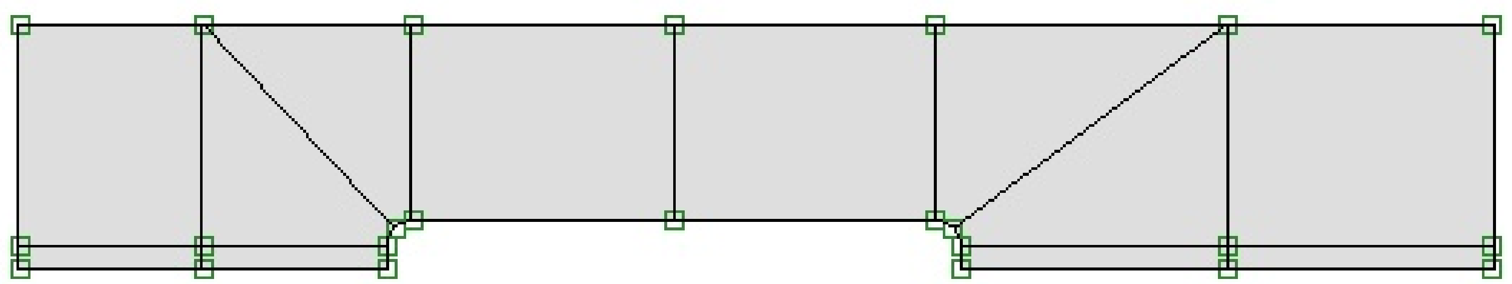
Manual construction

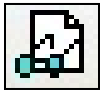
- Manual construction is recommended when a part is to be analyzed repeatedly, as in a design study.
  - The part is characterized by a set of parameters.
  - Example: See Exercise 4.3.9 in the text.



Manually constructed mesh

- Example of a manually constructed mesh
  - The elements and the nodes are shown below



  - The name of the input file is NotchedBeam.sci
  - To read the input file click on the icon  then select the filename and double click on it.
  - We will construct this mesh step by step.

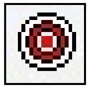
Enter the parameters

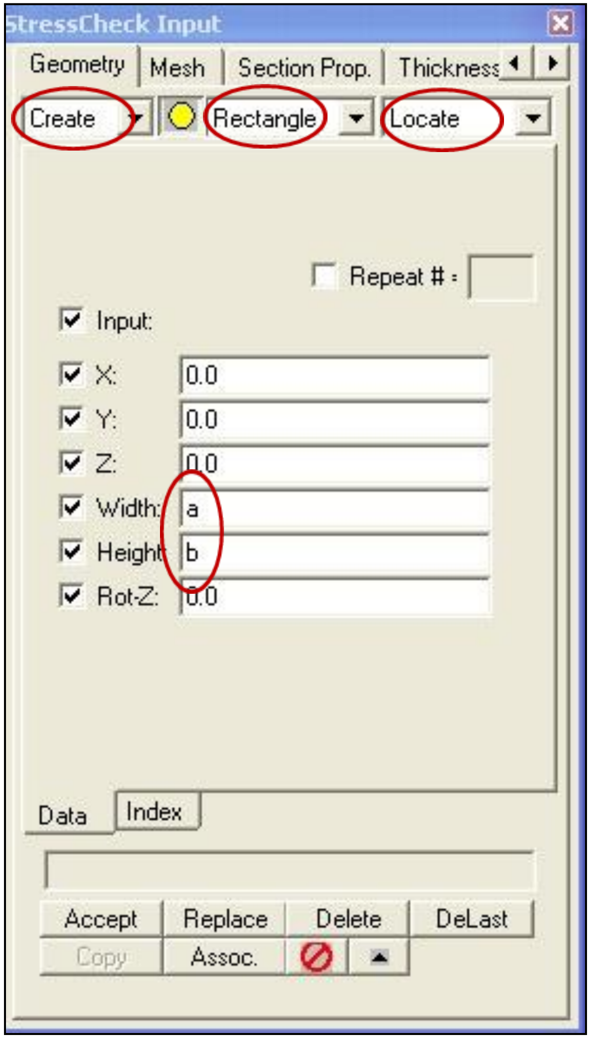
- Click on the icon  and select the parameter tab. Enter the parameters.
  - Note that M2 depends on M1 and V. – Why?

Model Info | Parameters | Rules

Name	Description	Expression	Value	Limit	Class	Sort
a	dimension		1.8000e+002		General	01
b	dimension		3.0000e+001		General	02
c	dimension		4.5000e+001		General	03
d	dimension		6.5000e+001		General	04
h	dimension		2.4000e+001		General	05
rf	radius of fillet		3.0000e+000		General	06
th	thickness		4.0000e+000		General	07
M1	moment		1.0000e+004		General	10
M2	moment	M1+a*V	5.5000e+004		General	11
V	shear force		2.5000e+002		General	12

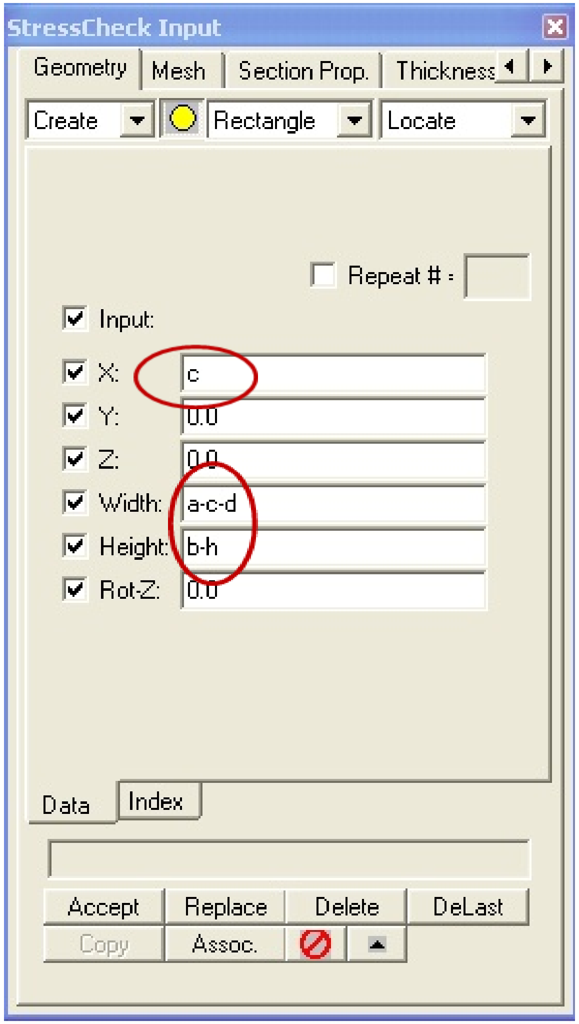
Construct geometrical objects (1)

- Click on the icon  and select the geometry tab
  - *Select Create > Rectangle > Locate*
    - Enter parameters a, b as shown.
    - Click *Accept,* and the rectangle will appear.
  - Center rectangle by clicking on the icon 
    - The global coordinate system is in the lower left corner of the rectangle.



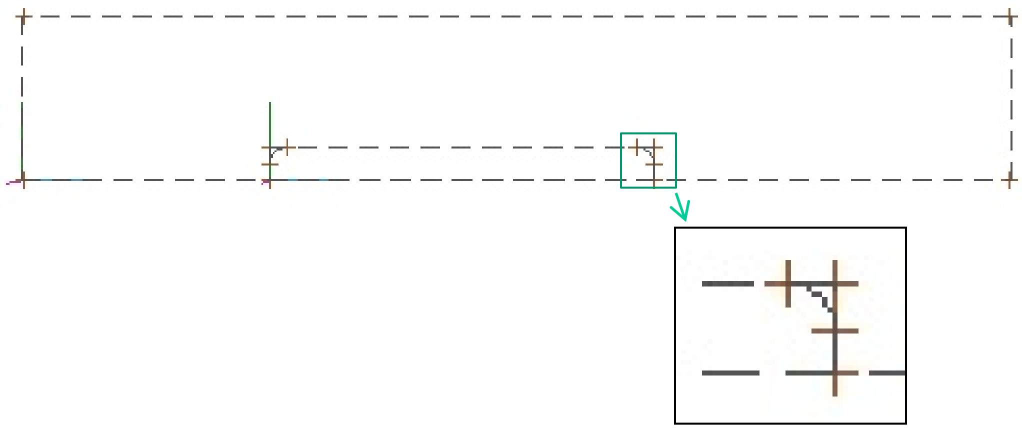
Construct geometrical objects (2)

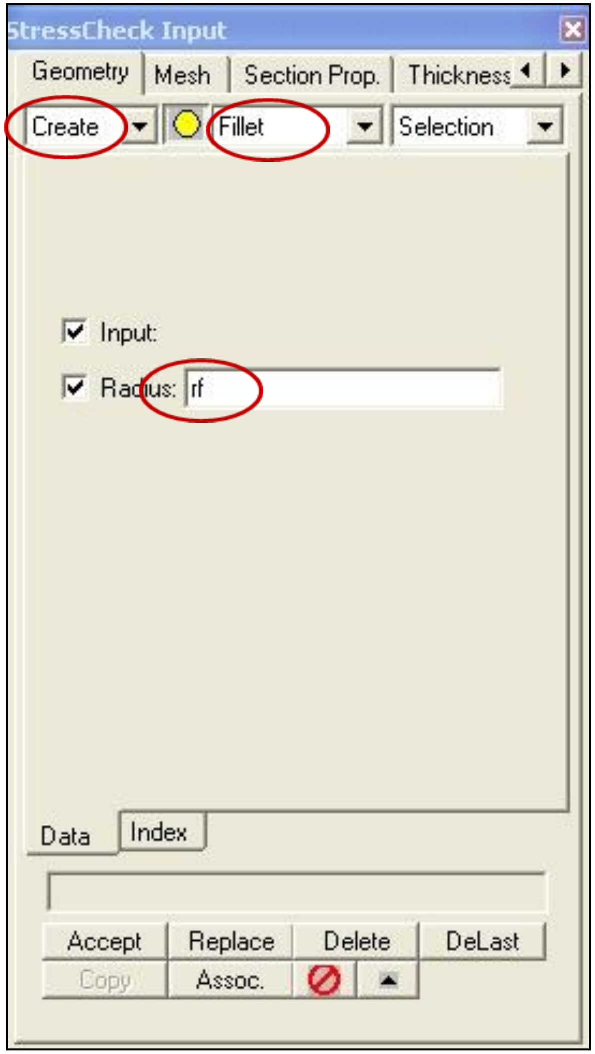
- Create a second rectangle using the parameters shown on the right.
  - The result is shown below.



Construct geometrical objects (3)

- Create fillets
  - Specify radius (rf).
  - Click on each of the intersecting lines where you want the fillet inserted.
  - Points are created at the ends of the circular arc.





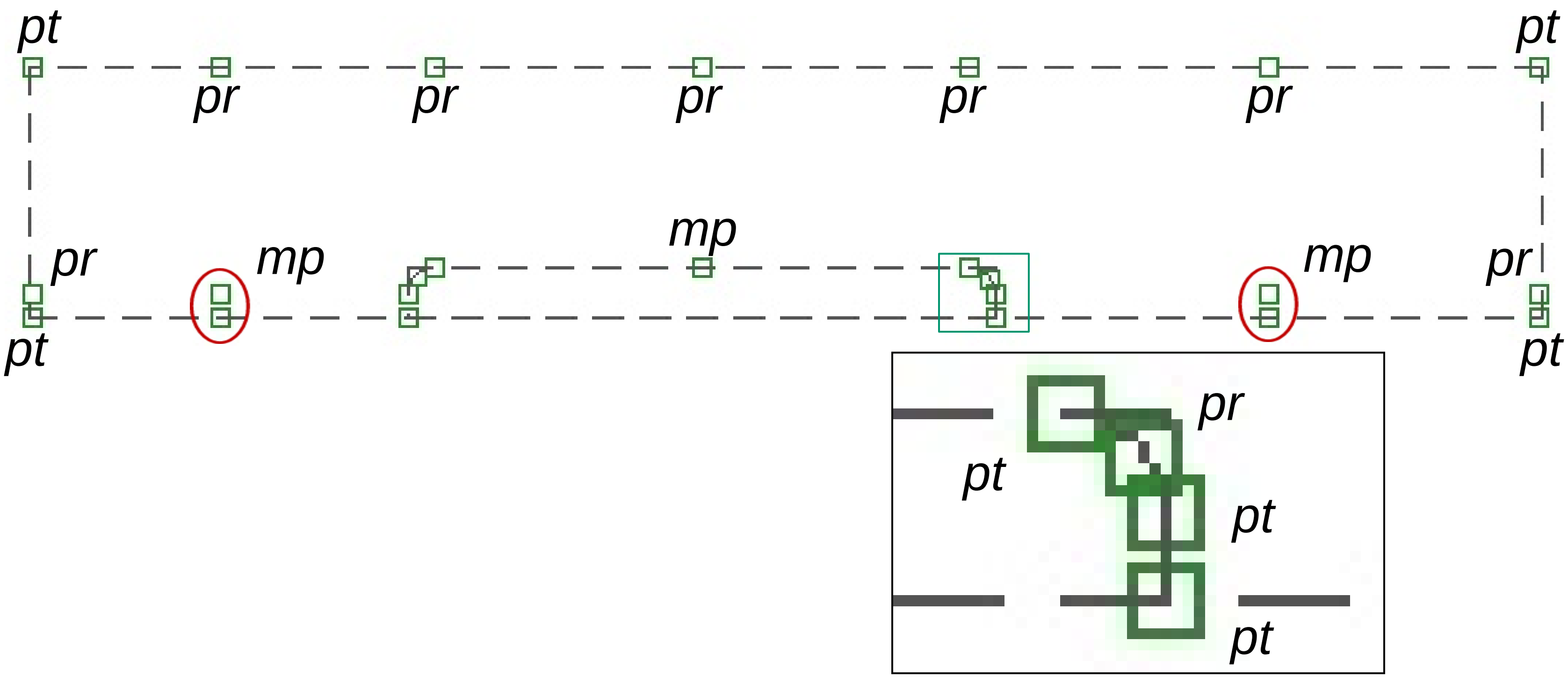
Construct mesh (1)

- To construct a mesh first create the nodes then create the elements. *Select Mesh*.
  - Nodes can be created in several ways. We will create nodes by
    - locating them where points already exist
      Create > Node > Point select the point(s) where you wish to locate nodes click accept
    - the mid-point method
      Create > Node > Midpoint click on the two nodes, a node will be created in the mid-point
    - projection

      Create > Node > Projection select the node(s) you wish to project. Hold the Ctrl and Shift keys then click Accept.

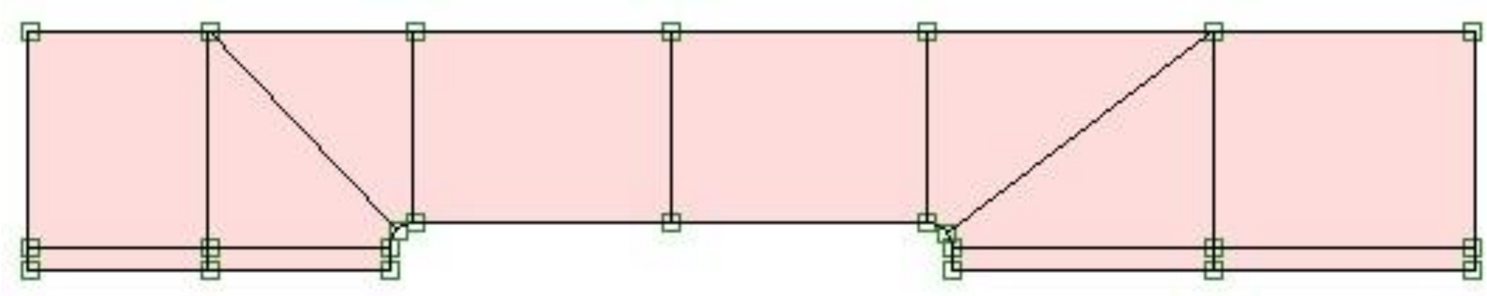
Construct mesh (2)

- After turning off the Point and System displays you should see the figure below.
  - The labels *pt*, *mp* and *pr* indicate the point, mid-point and projection methods



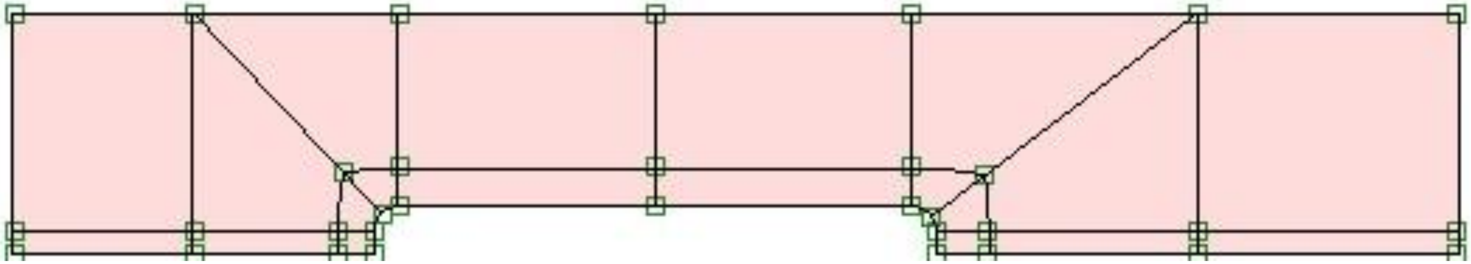
Construct mesh (3)

- Create elements:
  - *Create > Quadrilateral >Selection*
  - Click or select any four nodes. A quadrilateral element will be created.
  - Alternatively hold Ctrl - Shift then bring the cursor near the centroid of the element you wish to create. A green outline will appear. Left click and release Ctrl – Shift.

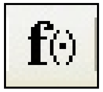


Check mesh & refine

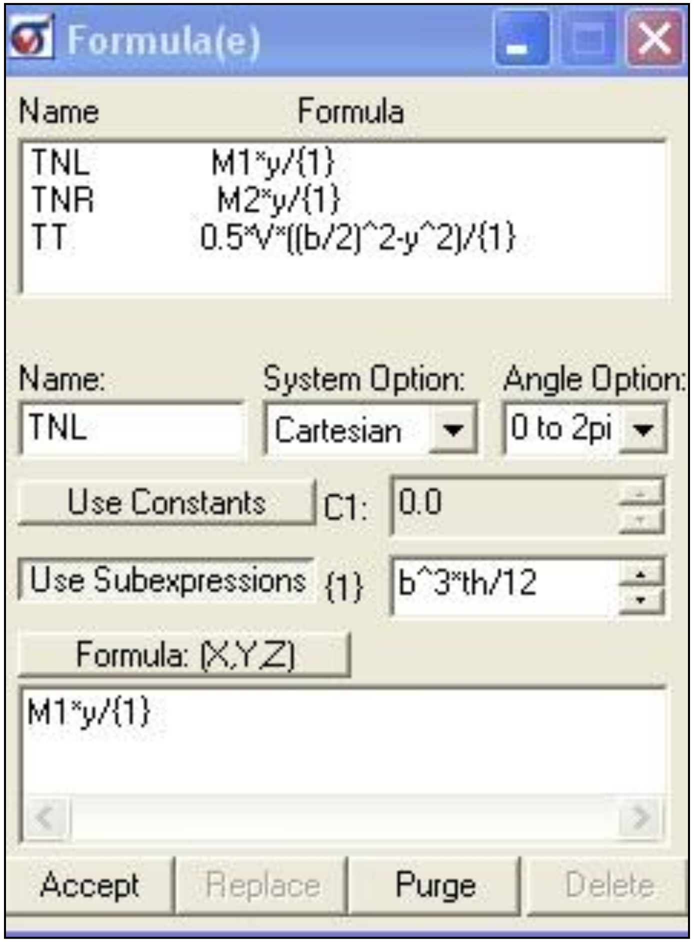
- Check for errors in meshing
  - *Check* > *Edge* > *Free Edge*. The free edges will be highlighted. Troubleshoot if necessary.
- Refine (read Example 7.3.1 in the text)
  - Click on the tab h-Discretization
  - *Select* > *Edge Curve* > *Simple Graded*
  - Name (any name), Layer (leave blank), Midsides (1), Grading (0.2)
  - Click on any edge on the cut-out, *Accept, Mesh*.



Define formulae

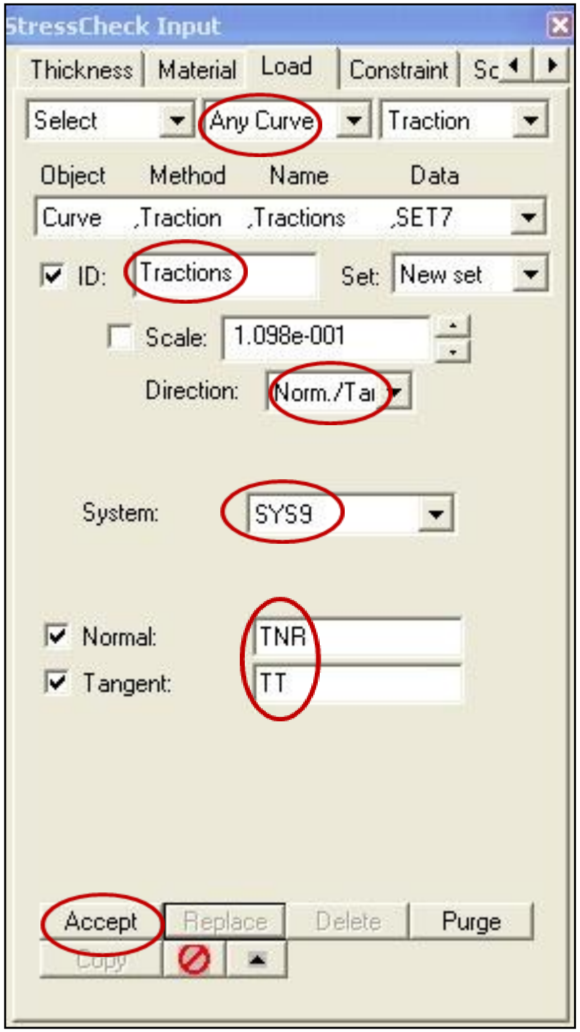
- Click on the icon 
  - Define formulae for the normal traction on the right (TNR), normal traction on the left (TNL) and the shearing traction (TT) as shown on the right.
- Create a centroidal coordinate system
  - Geometry > Create > System > Locate

    X = 0, Y = b/2, Z = 0



Apply load

- Click the Load tab
  - *Select > Any Curve > Traction*
  - *Click on the curve(s) to be loaded*
  - ID: Any name (*Tractions*)
  - Direction: Norm/Tan
  - System: Select the system created in the previous step
  - Normal: Formula (*TNR*)
  - Tangent: Formula (*TT*)
  - *Accept*

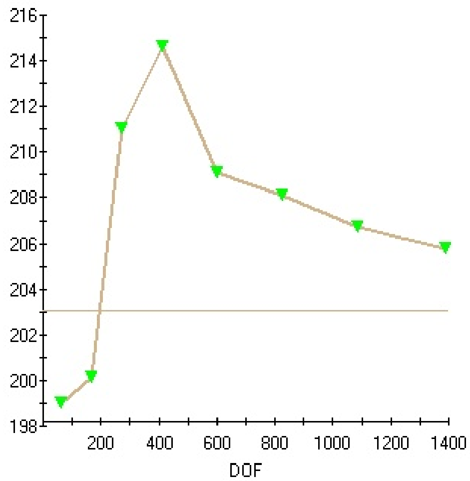


Check equilibrium

- Rigid body constraints should be applied only when the body is in equilibrium. – Why?
- To check whether the applied loads satisfy equilibrium;
  - *Check > All Elements > Selection > Accept*
  - A report showing the net force components and moment acting on the body will appear. These should be very nearly zero, that is, orders of magnitude smaller than max($|M_1|$, $|M_2|$, V).
  - If the body is not in equilibrium then there is an error in input. Trouble shoot.

Solution & post-processing

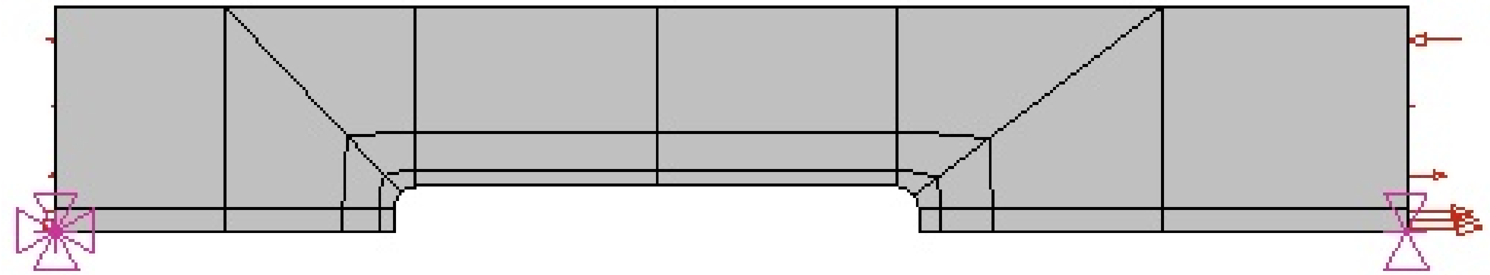
- Enter the material properties, apply rigid body constraints and create the solution record.
- Solve p = 1 to p = 8.
- Extract the first principal stress. You will see the results below. At p=8 S1 = 205.7 MPa



Run	DOF	Max. S1
1	63	1.9900E+02
2	167	2.0010E+02
3	271	2.1100E+02
4	415	2.1460E+02
5	599	2.0910E+02
6	823	2.0810E+02
7	1087	2.0670E+02
8	1391	2.0570E+02

Modifying the mesh

- Modify the record for h-Discretization so that Midsides = 2, Grading = 0.3 and click on Mesh. You will see the mesh shown below:



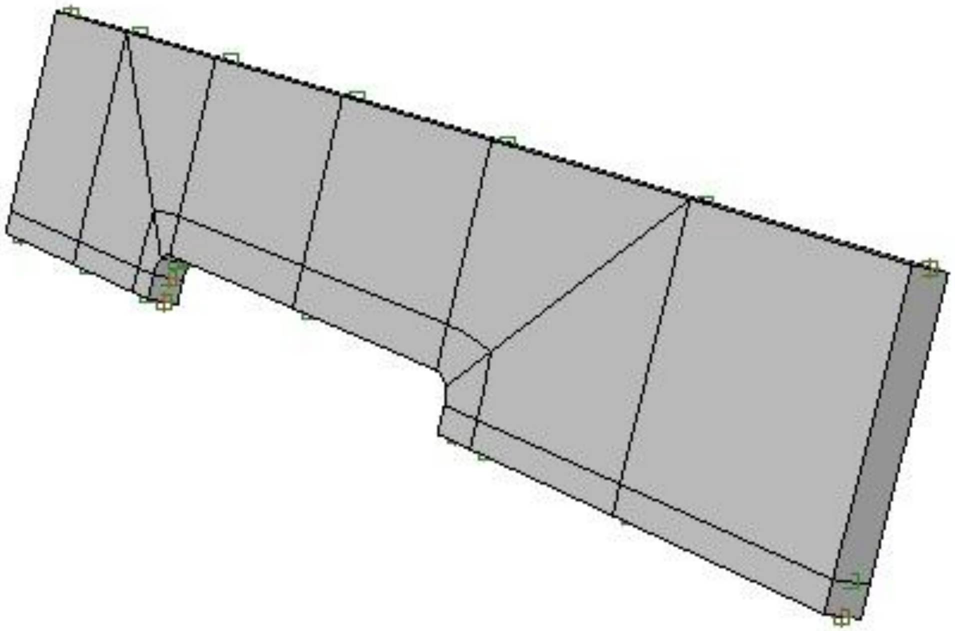
- Solve again and extract the maximum value of the first principal stress. At p = 8 you will find it to be 204. 5 MPa.

Extrude

- If you have access to the Professional Edition of StressCheck then you can convert this planar problem to a 3-dimensional one by extrusion.
- Let us impose constant moment on the boundaries BC and DA determine the value of the moment that will cause an out of plane displacement of 2.0 mm.

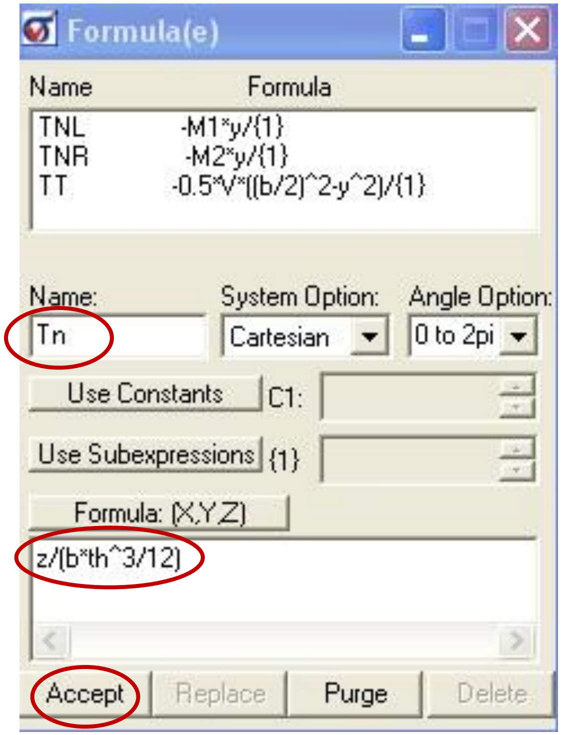
Extrusion

- Select Extrude (Reference/Theory toolbar).
  - You you will see the body shown below.



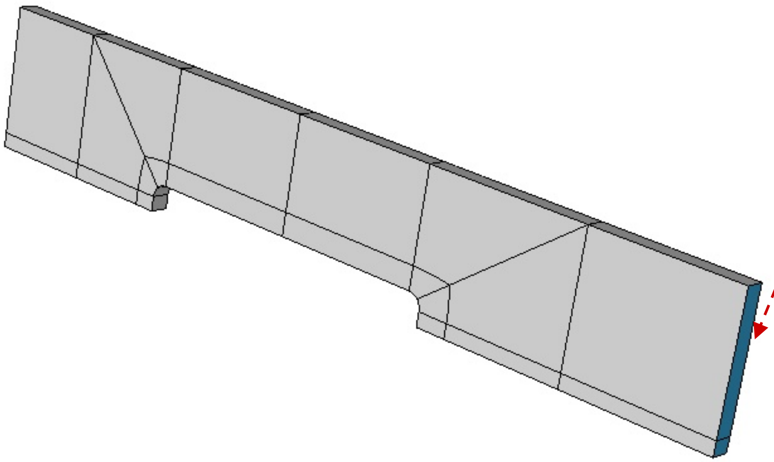
Define the normal traction

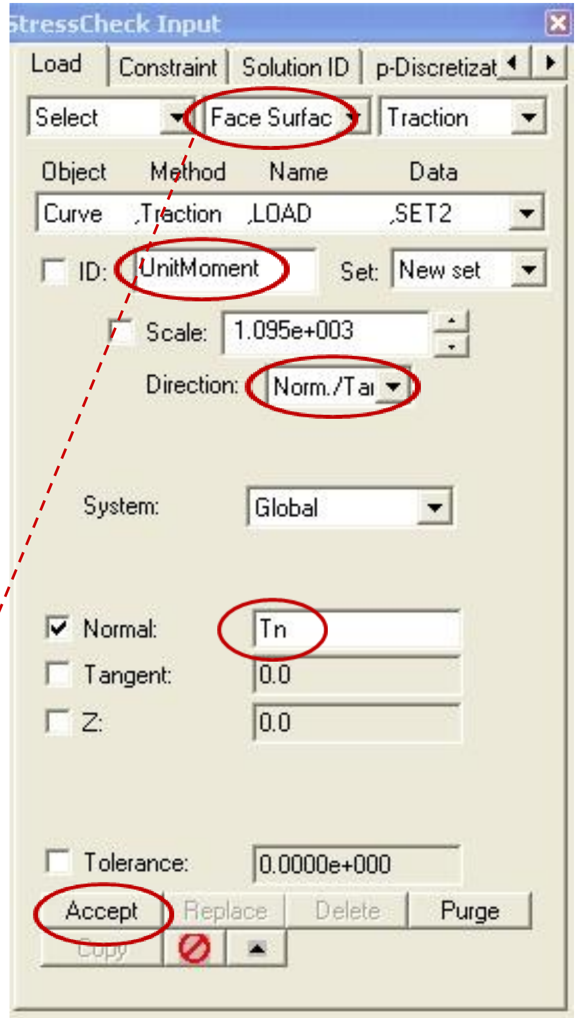
- Define formula for the normal traction T_n corresponding to a unit moment.
  - Enter the formula shown on the right.
  - Click *Accept*



Apply tractions

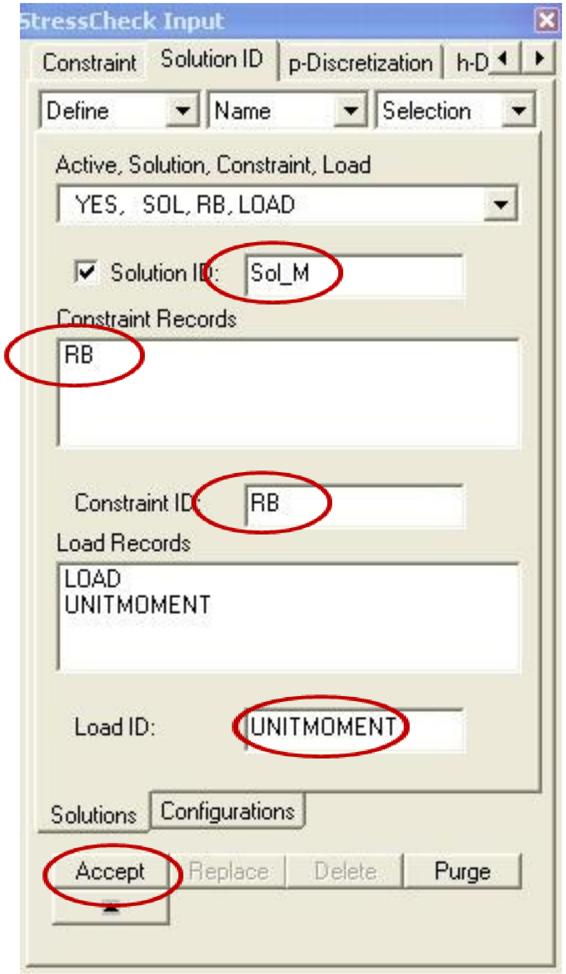
- Apply unit moment
  - Select Face Surface, enter ID (UnitMoment), Direction: (Norm/Tan), Normal: Tn *Accept*.
  - Repeat for the opposite side
  - Check equilibrium.





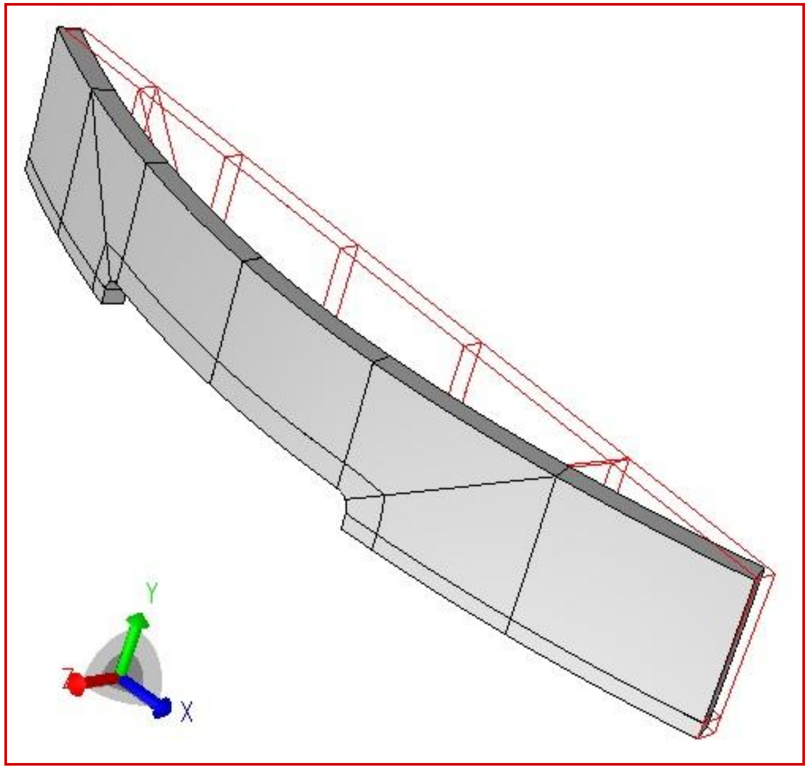
Create solution record

- StessCheck automatically converted the rigid body constraints to 3D.
- Create a new solution record
  – Solution ID (Sol_M)
  – Constraint Record (RB)
  – Load Record (UNITMOMENT)
- Solve
  – Since there are two active solution records, there will be two solutions.



Display the deformed configuration

- Plot the deformed configuration
- Find the maximum displacement Uz:
  - $(Uz)_{max}$ = 4.369E-4 mm
  - The moment that will cause 2 mm maximum displacement is therefore M = 4578 Nmm.



Recommended exercises

- Determine the maximum von Mises stress in the extruded configuration corresponding to the applied moment of 4578 Nmm and verify that the relative error is not greater than 2 %.
  - Answer: 119.8 MPa.
- Compare the maximum normal stresses computed for (a) the two dimensional body and (b) the extruded body loaded in-plane only.
  - Answer: (a) 204.5 MPa (b) 189.6 MPa
  - Note: For answer (a) see p. 31.
  - Note: The location of the maximum is the same for (a) and (b): x = 112.8 mm, y= 5.88 mm.

Recommended exercises

- Import the file NotchedBeam1.x_t
- Create the parameters

Model Info | Parameters | Rules

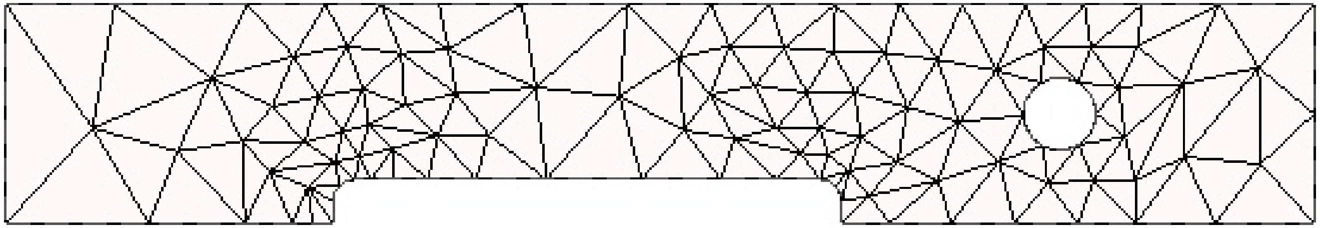
Name	Description	Expression	Value	Limit	Class	Sort
x0	coordinate		1.4500e+002		General	01
y0	coordinate		1.5000e+001		General	02
r0	radius		5.0000e+000		General	03

- Create a circle with center (x0,y0), radius r0
- Use Boolean subtraction to create the sheet body;

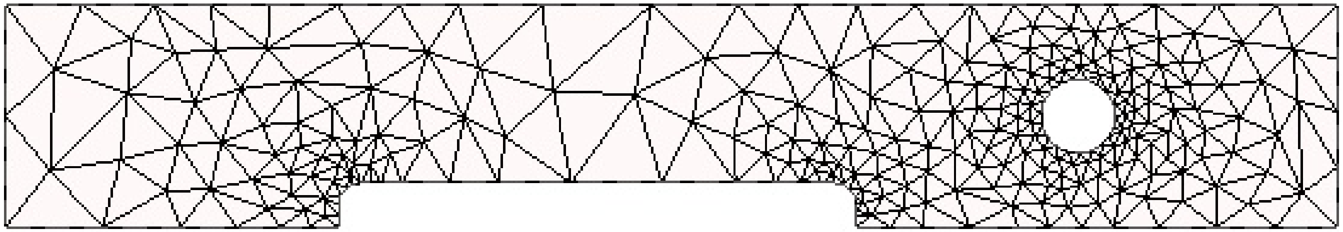
- Construct an isoparametric mesh (automesh).

Recommended exercises

- Using the default parameters you should see a mesh like this:



- Using the default refinement at the curved boundary segments, you should see a mesh like this:



- What will happen if you change the parameters that define the circular hole?