



Tutorials

Mecway Finite Element Analysis

Version 14.0
2021

Contents

Chapter 1

Getting Started 3

- 1.1 Quick Start, 3
- 1.2 Basic Operations in the Graphics Area, 4
- 1.3 Solution, 8
- 1.4 Manual Meshing, 10

Chapter 2

Analysis Types 16

- 2.1 Static Analysis of a Pressurized Cylinder, 17
- 2.2 Thermal Analysis of a Plate being Cooled, 22
- 2.3 Heat Flow Through a Solid Part, 26
- 2.4 Thermal Stress, 28
- 2.5 Free Vibration of a Cantilever Beam, 30
- 2.6 Dynamic Response of a Crane Frame, 34
- 2.7 Dynamic Response of a Solid Part, 37
- 2.8 DC Circuit, 39
- 2.9 Electrostatic Analysis of a Capacitor, 42
- 2.10 Acoustic Analysis of an Organ Pipe, 45
- 2.11 Buckling of a Column, 50

Chapter 3

Operations 54

- 3.1 Extrude, 54
- 3.2 Loft, 55
- 3.3 Refine Local 2D, 56
- 3.4 Mirror Symmetry, 57

- 3.5 Cyclic Symmetry, 58
- 3.6 CAD Work-flow, 59
- 3.7 Local Refinement with CAD Models, 61
- 3.8 Mixed Materials, 62
- 3.9 Mixed Materials with CAD Assemblies, 63
- 3.10 Constraint Equations, 65
- 3.11 Advanced Manual Meshing, 66
- 3.12 Assembly with Contact, 77

Getting Started

These tutorials assume that you are new to Mecway and indeed may be new to finite element analysis. Here you will find step by step instructions to get you started using the program. Once you have learned the basic concepts and operations, you can find more comprehensive information in the companion '*Manual*'.

1.1 Quick Start

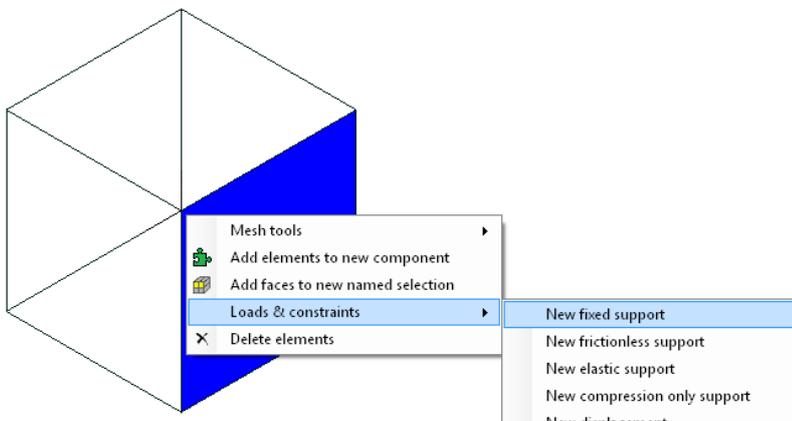
This is a very simple tutorial showing you how to create a working model from scratch. You can skip this if you prefer a more in depth introduction from the other tutorials.

Step 1

Change to Select faces mode. 

Step 2

Click the Quick cube button  to create a hexahedron element.

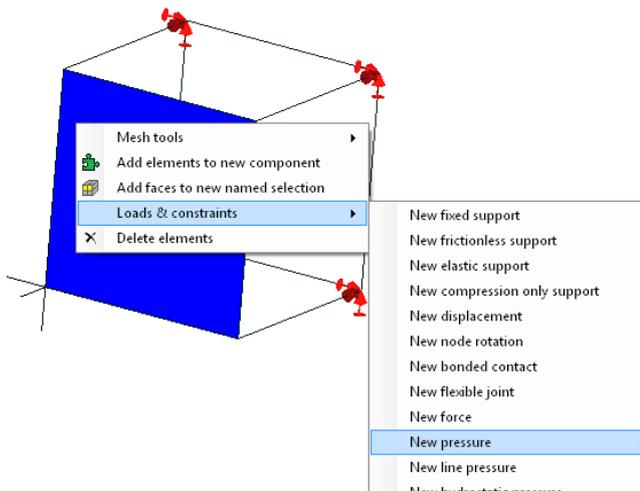


Step 3

Right click the right-most face and click **Loads & constraints** then **New fixed support**. Click **OK** in the box that appears.

Step 4

Drag with the middle mouse button to rotate the model so the opposite face is towards you. If you don't have a middle mouse button, use the Rotate tool button  and the left mouse button.

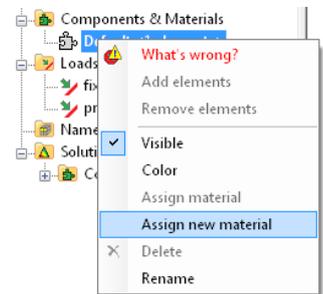


Step 5

Right click the opposite face and click **Loads & constraints** then **New pressure**. Enter the value **5** and press **OK**.

Step 6

Right click the **Default** component in the outline tree and click **Assign new material**. Choose **Isotropic** and enter a **Young's modulus** of 50000. Press **OK**.

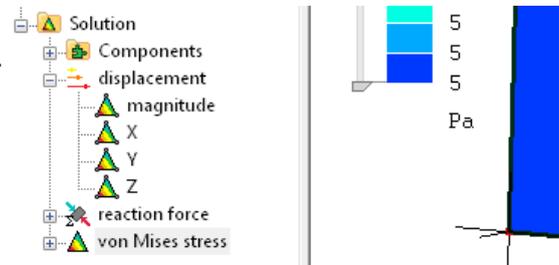


Step 7

Press solve. 

Step 8

Click the **von Mises stress** field variable in the outline tree. The color key shows a 5 Pa uniform stress due to the pressure load.

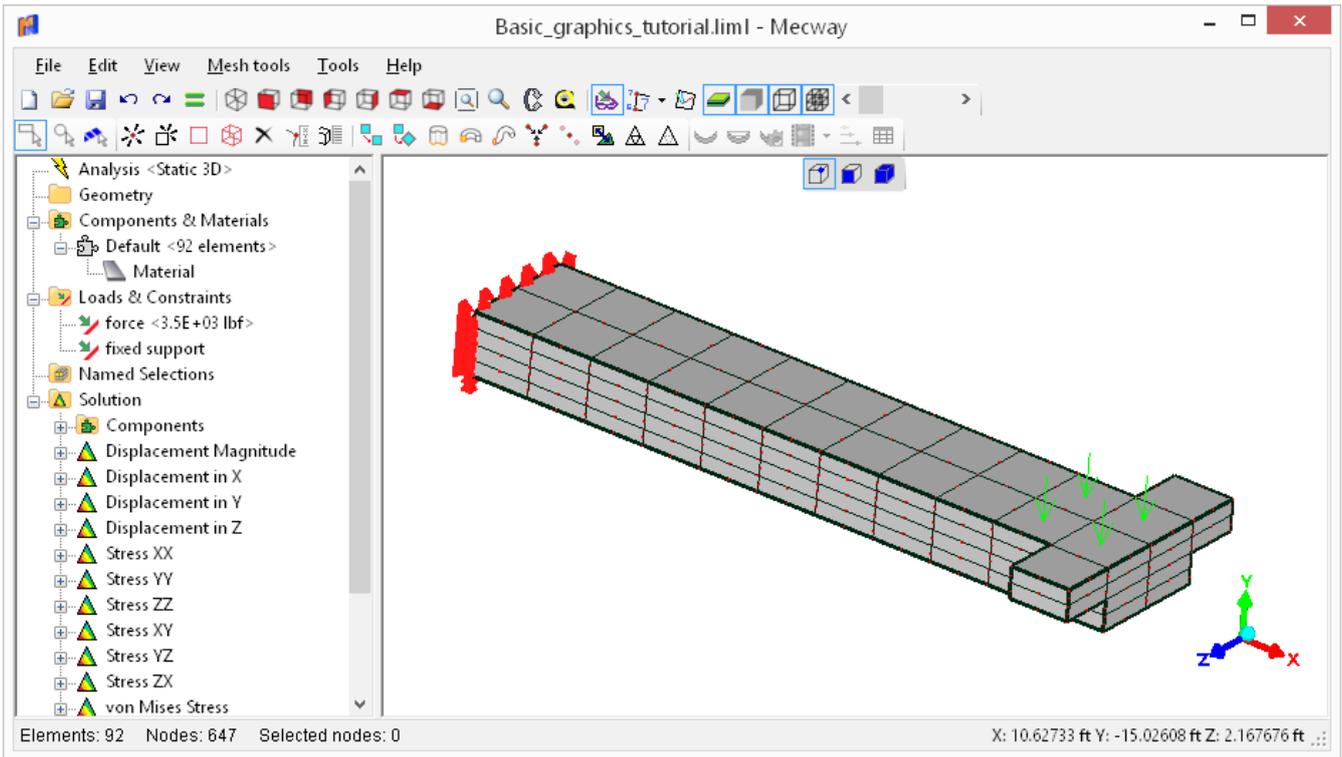


1.2 Basic Operations in the Graphics Area

Because the graphics is so much a part of Mecway, let us look first at a data file which displays a model and its solution.

Step 1

File → **Open**, **Basic_graphics_tutorial.liml** in the tutorials folder where Mecway has been installed. This is a contrived example merely to show the graphics features.



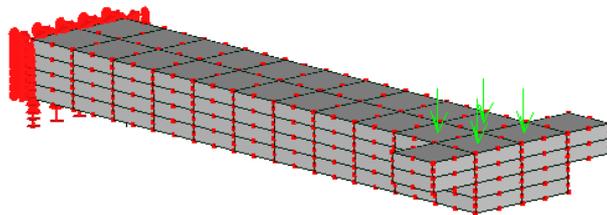
There are three parts to the screen

- the toolbars at the top, arranged into model-building tools and the graphics display options.
- the model structure and solution displayed in an outline tree in the left panel
- a graphics area which displays the mesh

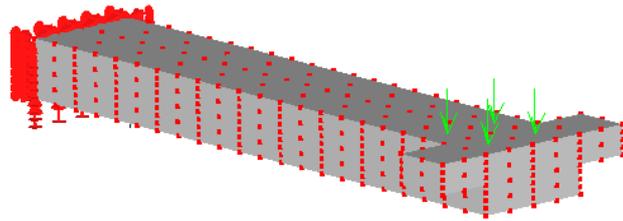
This sample is a beam with a T-shaped end and has already been solved. One end is totally fixed, as if built into a wall. At the free end, a downwards force is applied at 4 faces. The self-weight of the beam has been neglected.

Step 2

 Toggle the display of element surfaces.



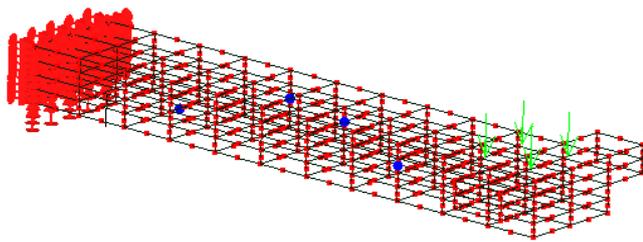
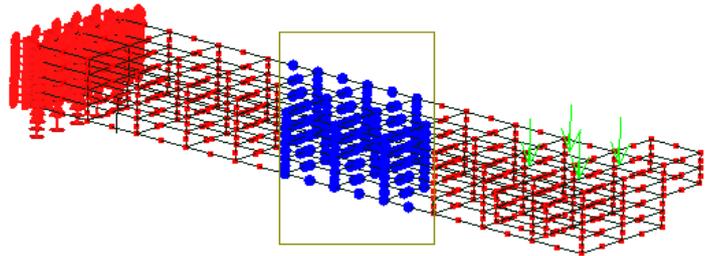
 Toggle element edge display.



Step 3

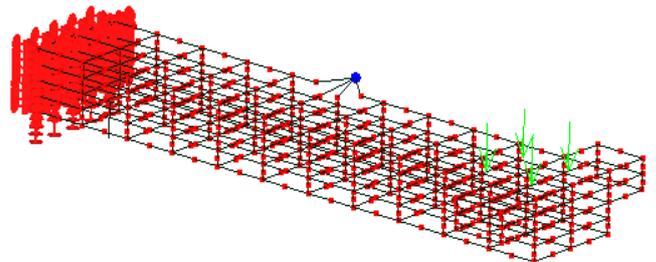
 Select nodes

Turn off the display of elements surfaces and turn on element edges. Drag over the model to select nodes, including internal nodes.

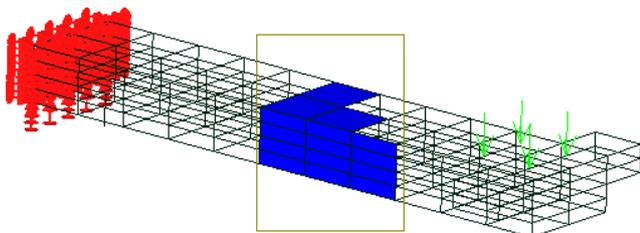


Click in an open space to deselect the nodes. Then click a node to select it. Hold the **Ctrl** key down and click a couple of nodes to add to the selection set. If a node is already selected and it's clicked on while holding down the **Ctrl** key, it becomes deselected.

Drag a node to a new location. Repeat the action on another node, but this time hold down the **Shift** key. You'll notice that the node is fixed and cannot be dragged.

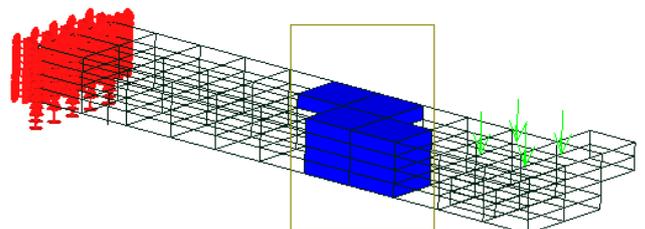


Edit → **Undo** or **Ctrl + Z** to return the displaced node.



Click in an open space to deselect the nodes. Select faces . Drag over the model to see that only surfaces are selected.

Click in an open space to deselect the surfaces.

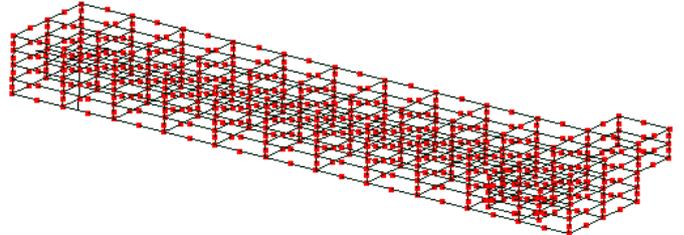


Select elements  . Drag to see that only elements have been selected.

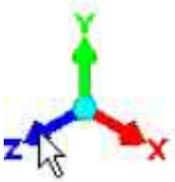
The **Ctrl** key has the same effect while selecting faces or elements as it does with selecting nodes.

Step 4

 The display of loads and constraints can be toggled off or on.

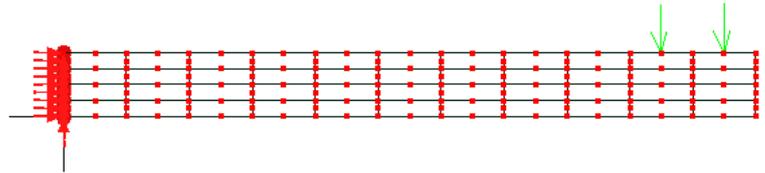


Step 5

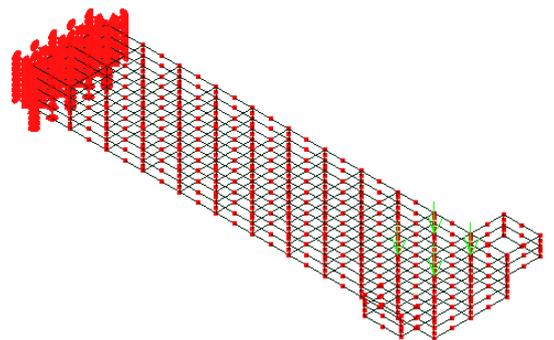


Click the Z arrowhead of the triad at the bottom right corner of the graphics area to view the model parallel to the screen.

Left or right clicking the arrowheads will display the different views of the model parallel to the screen.



Click the blue dot to return to an isometric view.



Step 6

To rotate the view, drag with the middle mouse button (or wheel).

To see a small feature more clearly, zoom into that area by placing the mouse over that area (no clicking required) and rotating the mouse wheel for a larger display of the small feature. Rotating the mouse wheel the other way will make the model's display become smaller.

If the model's display seems to be half out of the graphics area, you need to pan the model back into view. Drag it using the right mouse button.

If the model does not fully appear in the graphics area after you have used the zoom, rotate or pan, use the fit to window tool-button .

Step 7

Use the tape measure tool-button  to check the lengths. Click a node but continue keeping the mouse button pressed and move  the cursor across to another node. Mecway will give a readout of that distance.

Step 8

Tools → **Volume** will give volume of the entire mesh. This tool can be used to obtain the volume of a selected part of the mesh. The selection can be element nodes, faces or elements.

Similarly the **Tool** → **Surface area** will give the area of the selected faces. The selections can only be element faces.

1.3 Solution

Step 1

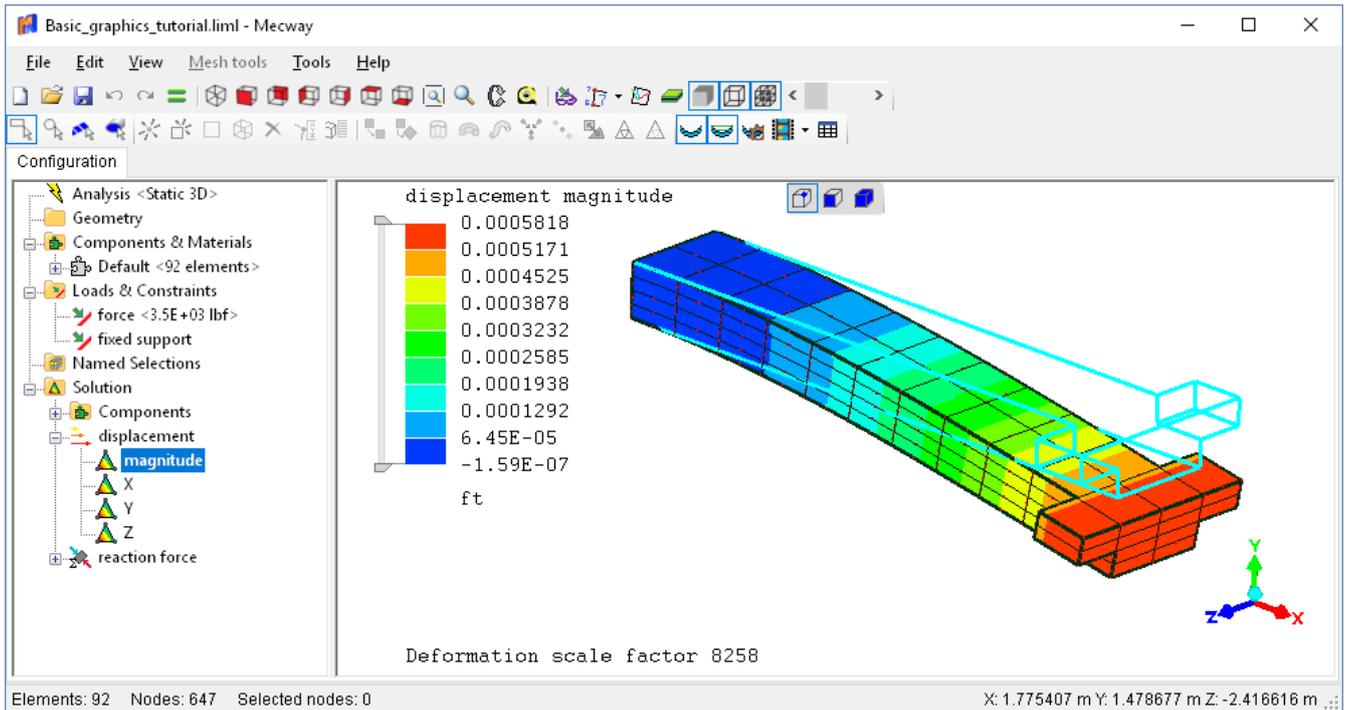
This tutorial uses the same file as the previous tutorial. So if it is not already open in Mecway, use the **File** → **Open**, **Basic_graphics_tutorial.liml** in the tutorials folder where Mecway has been installed.

In the solution section of the outline tree click on **displacement** → **magnitude**. The graphics display will update to show the results of the solved model. The units used in the legend on the display scale can be changed by clicking on them.

Step 2

Click the  **Deformed view** tool-button to visualize an exaggerated displacement of the structure.

Click the  **Undeformed shape** to superimpose an outline of the undeformed shape onto the deformed mesh.



Step 3

Right click **Solution** in the outline tree and choose **new stress and strain** → **stress** to add the 6 stress components to the solution.

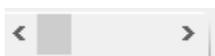
Step 4

Click the  table tool-button to display the displacements, rotations and stresses at each node in spreadsheet-style cells which you can then copy and paste into your own spreadsheet.

Step 5

Use the  animation tool-button to animate the deflection. Smooth deformation between the two extremes of movement is simulated. You can choose the scale factor. This tool is especially useful for visualizing solutions in vibration analysis.

Step 6



This slider can be used to cut-away the model to look inside it. You will need to rotate the model suitably so that the cut occurs where you want it. Don't forget to return to its leftmost position before doing any editing of the mesh.

To return to the model that does not display the results, click any of the items in the outline tree that is not below **Solution**.

Now that you have some experience in manipulating the graphics of the model, it's time to learn how to create them from scratch. The next section will walk you through the case we have just studied.

1.4 Manual Meshing

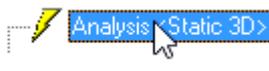
Always begin a manual mesh by creating a coarse mesh; it can always be refined later. A coarse mesh simply means larger and fewer elements, and a refined mesh means smaller and more elements. Creating a coarse mesh requires less labor and, if things go wrong, it will be less frustrating.

Just as in the real world where everything has three dimensions (length, width, height), the geometrical properties of finite elements are also three dimensional in nature. Some elements will appear on the screen as being clearly three dimensional elements, while others will appear on the screen as flat and two dimensional. Nevertheless, the elements that appear flat and two dimensional do actually have the third dimension, of thickness.

Elements that appear three dimensional on the screen will usually be created from a two dimensional flat shape, so modeling typically starts with what appears on the screen as a flat two dimensional mesh. This initial 2D mesh can be created either by a combination of nodes and elements or by using ready-made template patterns. Editing tools are available for modifying the two dimensional mesh as you create and form it. Once the coarse mesh is complete, whether it be two dimensional or three dimensional in appearance, it will need to be refined before running the solver.

Here is a tutorial to illustrate how the manual meshing tools work together to create the model used in the introductory chapter. We will recreate the T-shaped thick beam used in section 1.2 to illustrate Mecway's graphics tools. At the end you may wish to use some of the skills you have learned to modify the length and thickness of the beam to make it more realistic.

Step 1

 The analysis type should be **Static 3D**. If it's not, right click the item, then select **Analysis settings** to change the analysis type.

Use the **Mesh tools** → **Create** → **Node...**  or and enter the following coordinates.

X 0
Y 0
Z 0

Leave the units as meters (m).

Click the **Add** button. Repeat for the following coordinates, then click **Close**.

11,0,0
11,1,0
0,1,0

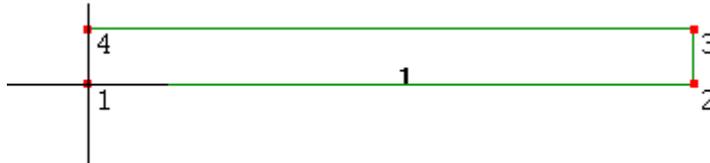
Click the Z arrowhead  to view the XY plane parallel to the screen.

To view the entire model, use fit to window 



Step 2

Mesh tools → **Create** → **Element...** quad4 shell Select and click the four nodes then click **Close**. The order of the clicked nodes will affect the orientation of the mesh refinement that will be done in the next step. In this tutorial, the element is formed using the node order 1-2-3-4.



Step 3

Mesh tools → **Refine** → **Custom...**

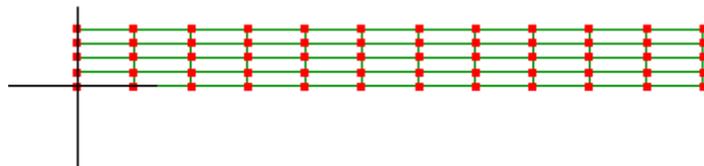
Number of subdivisions

R 11

S 4

T 1

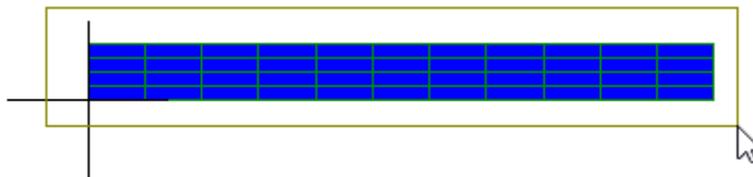
Click **OK**



Step 4

Activate select faces 

Drag to select the entire mesh.

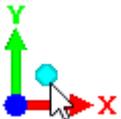


Mesh tools → **Extrude...**

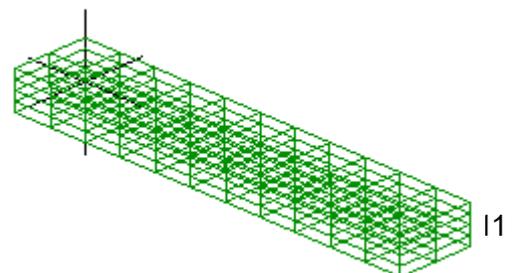
Direction +Z

Thickness 2, units m

Number of subdivisions 2



Click the blue dot to view an isometric display of the model.



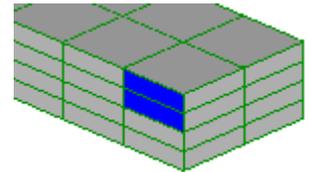
Click in the open space of the graphics area to deselect the elements.

Step 5

Activate select faces 

To improve clarity use the show element surfaces tool-button  to hide the internal elements.

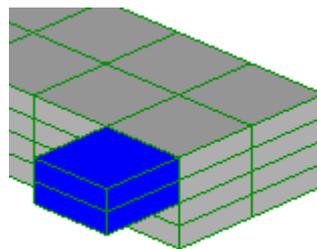
Select these two faces by clicking one face then holding the **Ctrl** key while clicking the second face.



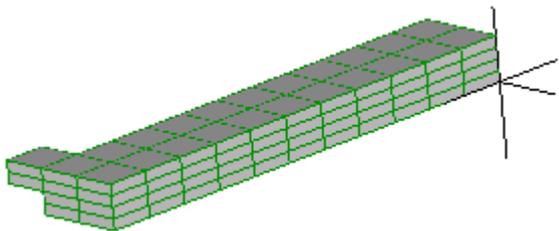
The **Ctrl** key can be used while selecting items. It works by adding the new items to the currently selected items, and if the clicked item is already selected it will become deselected.

Mesh tools → **Extrude...**

Direction +Normal
Thickness 3, unit ft
Number of subdivisions 1



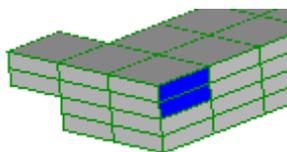
Notice that you can use a mixture of different units for different quantities.



Drag with the middle mouse button or wheel to rotate the view of the model or use this  if you don't have a middle button.

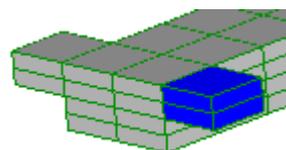
Activate select faces 

Select these two faces.



Mesh tools → **Extrude...**

Direction +Normal

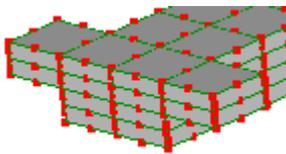
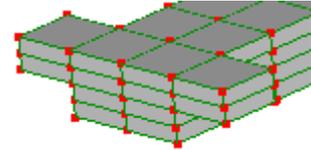


Thickness 3, unit ft
Number of subdivisions 1

Step 6

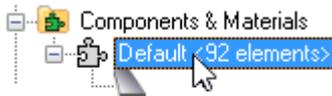
Click in an open space of the graphics area to deselect the model.

Activate select nodes  to see that these are 8 node hexahedrons.



Change these 8 node hexahedrons into the more accurate 20 node hexahedrons using **Mesh tools** → **Change element shape...** select **hex20** and click OK to accept. The hex8 elements give a linear approximation to the displacement field, whilst the hex20 elements give a quadratic approximation.

Step 7



Right click, **Assign new material**

Mechanical tab
Isotropic select
Young's modulus 200, unit GPa
Poisson's ratio 0.3

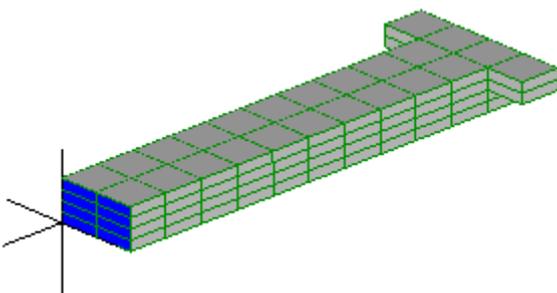
A material is now associated with the elements.



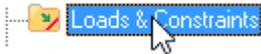
Step 8

In order for a part to develop stresses, all rigid body motion must be resisted. The left face of this model will be constrained.

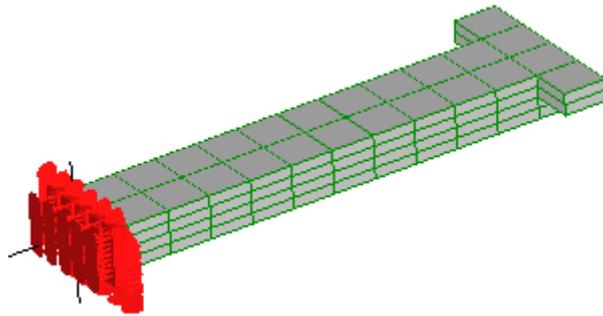
Activate select faces 



Select these faces. Hold the **Ctrl** key down while clicking to add the faces so that the faces already selected don't become deselected.



Right click, select **New fixed support** accept the defaults and click OK

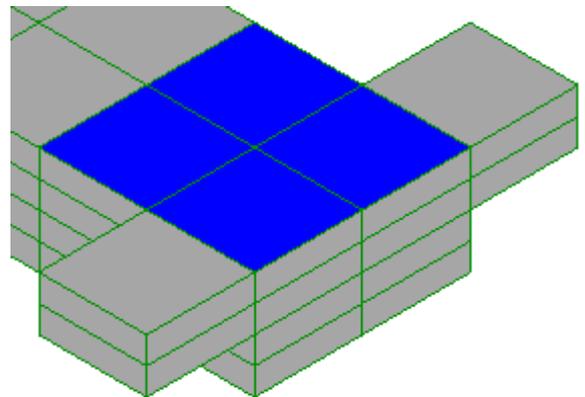


Step 9

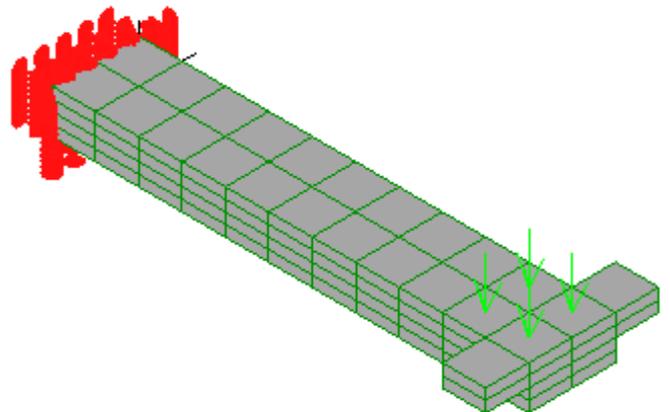
Select these faces.



Right click, select **New force**
Y -3500, unit lbf



The 3500 pound force will be distributed uniformly over the selected faces. So each equally-sized face will have a load of $3500 \text{ lbf} / 4$ or 875 lbf.

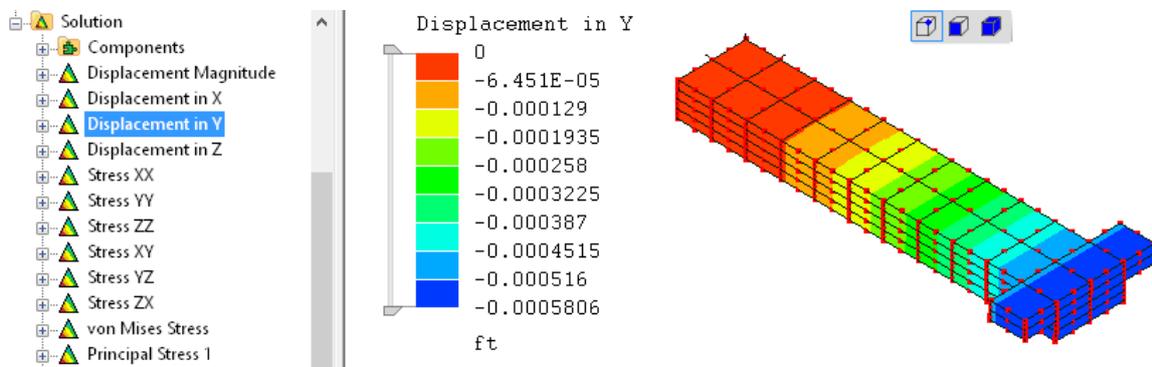


Step 10

Before you run the solver, check down the outline tree to make sure there are no warnings in red.

Click  to solve the model.

The results are listed in the outline tree below **Solution**.



You may now wish to explore Mecway's mesh-modifying tools, for instance by scaling the length, width and thickness of the beam using **Mesh tools** → **Scale...** or changing the applied load and/or elastic constants by right clicking to edit the appropriate item in the outline tree.

The next chapter is a step-by-step walk through for some of the various analysis types that can be modeled in Mecway.

Analysis Types

While the typical user does not need an in-depth study of the mathematics behind finite element analysis, you do need to understand the behavior of elements in order to represent a given physical problem correctly.

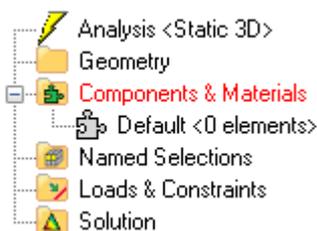
Finite element analysis is not like CAD (computer aided design) software where you simply create a geometry and take a print-out. Instead, it follows the law of 'Garbage in, Garbage out'. Your choice of element type, mesh layout and constraints will strongly affect the accuracy of the solution.

We recommend to beginners that you confine your models to text-book problems with known solutions rather than attempting real world problems with unverifiable solutions. When you get to the point where you're solving real world problems, never accept the results at face value. Rather, validate them by comparing the results to hand calculations, experimental observations or knowledge from past experience.

This chapter contains tutorials to initiate you into using Mecway's basic analysis capabilities. Some of the more specific features like cyclic symmetry and constraint equations are described in Chapter 3 - *Operations*.

Not all types of elements can be used for all types of analysis. The *Manual* lists the elements that are available for use in each type of analysis. When creating elements, if you see an N/A ('not applicable') next to the type of element, it means it can't be used to solve that type of analysis. However, you can use non-applicable elements as construction tools provided you change them using **Mesh tools** → **Change element shape...** , or delete them before solving.

line2	beam/truss
line3	N/A
tri3	N/A
tri6	shell
quad4	shell



The outline tree presents all the information you need about your model and allows you to perform various actions on the model itself. You will always begin at the top, changing the analysis type if you do not want the default, 3D static analysis.

Items that appear in **red** indicate missing or erroneous information, so right click them for a **What's wrong?** clue.

The constraints and loads that you apply will be listed in the **Loads & Constraints** section. After solving the model the field values will be listed below **Solution**.

2.1 Static Analysis of a Pressurized Cylinder

A cylinder of 2m radius, 10m length, 0.2m thickness, Young's modulus 200 GPa and Poisson ratio 0.285 will be analyzed to determine its hoop stress caused by an internal pressure of 100N/m².

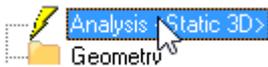
From shell theory, the circumferential or hoop stress for a thin cylinder of constant radius and uniform internal pressure is given by :

$$\sigma = (\text{pressure} \times \text{radius}) / \text{thickness}$$

$$\sigma = (100 \times 2) / 0.2$$

$$\sigma = 1000 \text{ N/m}^2$$

Step 1



Check that the default analysis type is Static 3D.

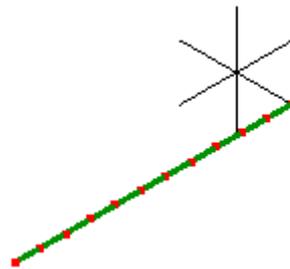
Step 2

Mesh tools → **Create** → **Create curve generator...** \ **Straight line**

select

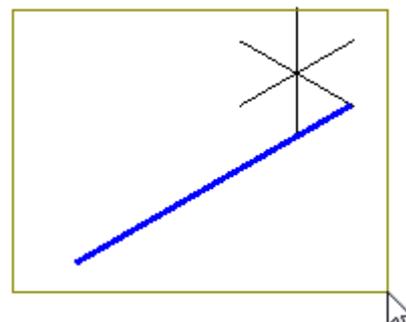
X1	2 m
Y1	0
Z1	0
X2	2 m
Y2	0
Z2	10 m
Number of nodes	12

Use the **Fit to window**  to display the elements.



Step 3

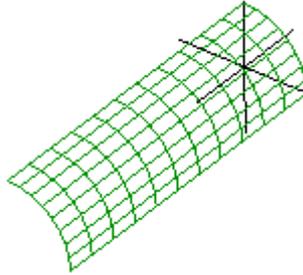
Due to the axial symmetry, only one quadrant will be



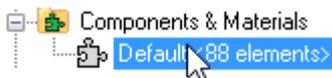
modeled.

Activate select faces  and drag to select all the elements.

Mesh tools → **Revolve...**
Axis of revolution +Z
Angle 90 °
Number of subdivisions 8



Step 4



Right click, **Assign new material**
Geometric tab
select **Shell/membrane**
Thickness 0.2 m

Mechanical tab
select **Isotropic**
Young's modulus 200E9 Pa
Poisson's ratio 0.285

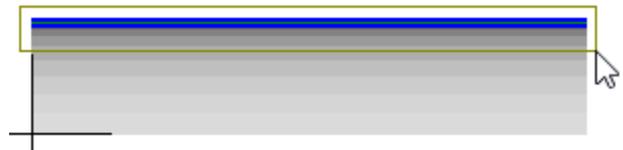
Step 5

Right click the X arrowhead  to view the YZ plane parallel to the screen.

Activate select faces,  activate show element surfaces,  and activate show shell thickness 

Because of the mirror symmetry only one quadrant of the cylinder has been modeled. At the planes of mirror symmetry, the nodes must be constrained so that they do not move out of the plane. Also, no bending must occur in that plane of symmetry.

To enforce mirror symmetry at the edge in the YZ plane, drag to select the edge of the shell.



 **Loads & Constraints** Right click, **New displacement** and select the **X** option.

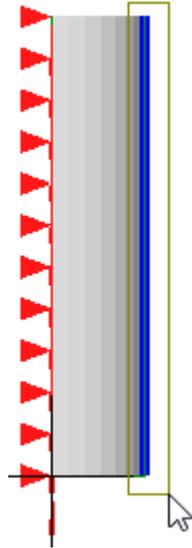
 **Loads & Constraints** Right click, **New node rotation** and select the **Z** option.

Right click the Y arrowhead  to view the ZX plane parallel to the screen.

To enforce mirror symmetry at the edge in the ZX plane, drag to select the thickness of the shell.

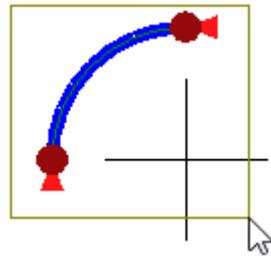
 **Loads & Constraints** Right click, **New displacement** and select the **Y** option.

 **Loads & Constraints** Right click, **New node rotation** and select the **Z** option.

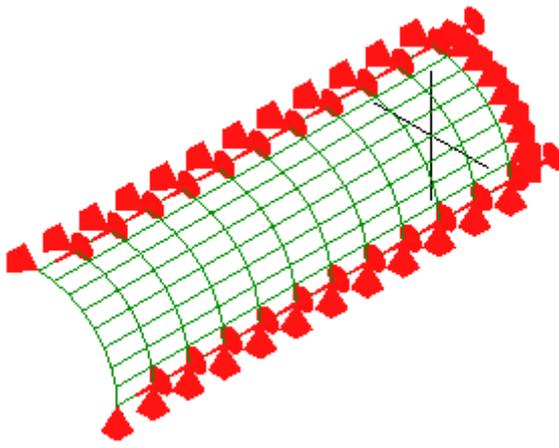


Right click the Z arrowhead  to view the XY plane parallel to the screen.

To eliminate rigid body translation motion along the Z axis drag to select the shell thickness in the XY plane.



 **Loads & Constraints** Right click, **New displacement** and select the **Z** option.

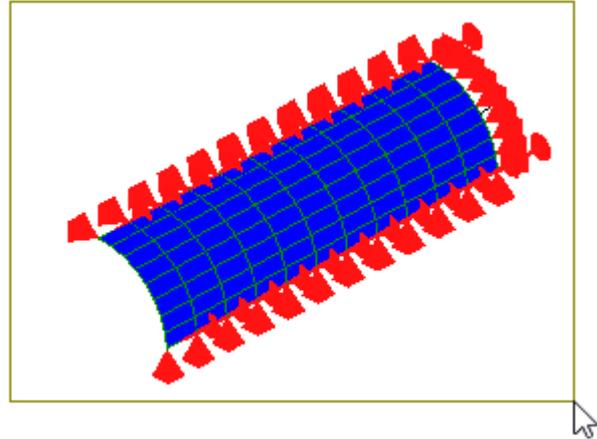


Step 6

Disable Show shell thickness  then drag to select the entire external surface.



Right click, **New pressure**
-100 Pa



Step 7

Click  to solve the model.

The results are listed in the outline tree below **Solution**. Click **von Mises stress** to view the hoop stress.

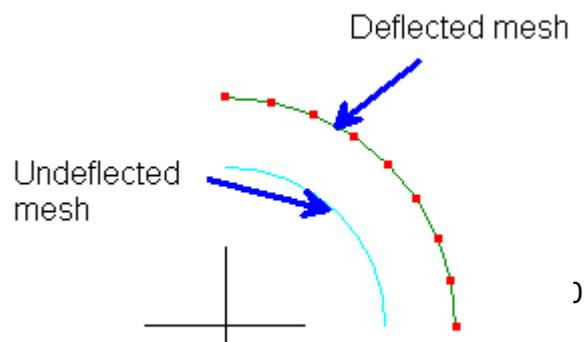
The first thing to check is the deformation. Errors in the applied constraints or loads can show up in the deformation shape.

Click the Z arrowhead  to view the XY plane parallel to the screen.

Click the **Deformed view** tool-button .

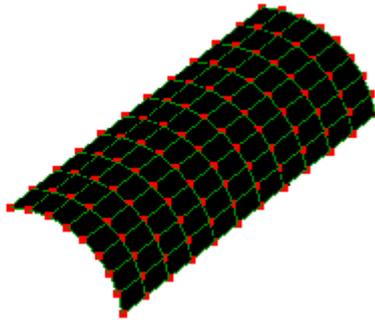
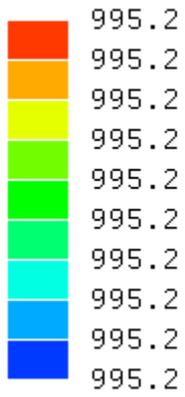
Click the **Undeformed shape** tool-button  to superimpose the undeformed geometry onto the deformed geometry.

Observe that the nodes in the YZ and ZX plane remain in these planes and no bending has occurred, this indicates that the constraints applied to enforce symmetry are working. The fact the deformation expands outward radially indicates that the internal



pressure has been correctly applied.

von Mises Stress, Membrane



The computed hoop stress is only 0.48% different from the hand calculations. This is close enough not to need further mesh refinement.

2.2 Thermal Analysis of a Plate being Cooled

A plate of cross-section thickness 0.1m at an initial temperature of 250°C is suddenly immersed in an oil bath of temperature 50°C. The material has a thermal conductivity of 204W/m/°C, heat transfer coefficient of 80W/m²/°C, density 2707 kg/m³ and a specific heat of 896 J/kg/°C. It is required to determine the time taken for the slab to cool to a temperature of 200°C.

For Biot numbers less than 0.1, the temperature anywhere in the cross-section will be the same with time. A quick calculation shows that this is true.

$$Bi = hL/k = (80)(0.1)/(204) = 0.0392$$

The 4 node quadrilateral element interpolates temperature linearly, and is able to represent unsteady states of heat transfer so this element will be selected for the model.

We need to have a rough estimate of the time required to reach a temperature of 200°C. In this case, from classical heat transfer theory the following lumped analysis heat transfer formula can be used.

$$(T(t)-T_a)/(T_o-T_a) = e^{-(mt)}$$

T_a = temperature of oil bath

T_o = initial temperature

where $m = h/ \rho C_p(L/2)$

h = heat transfer coefficient

ρ = density

C_p = specific heat

L = thickness

$$m = 80/[(2707)(896)(0.1/2)]$$

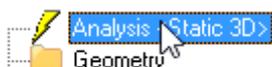
$$m = 1/1515.92 \text{ s}^{-1}$$

$$(200 - 50) / (250 - 50) = e^{-(1/1515.92)t}$$

$$t = \ln(4) \times 1515.92$$

$$t = 436 \text{ s}$$

Step 1

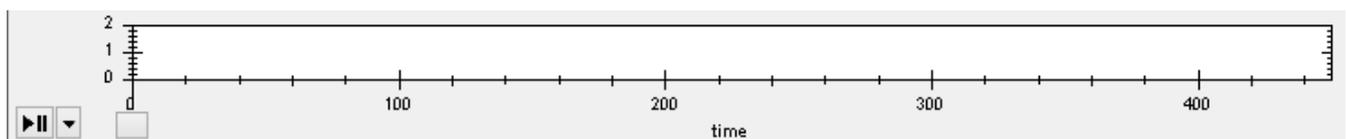


Right click, **Edit**. Select **Thermal Transient**

Time period 450 s

Time step 1 s

A slider is displayed to show the duration of the analysis, which in this case is 450 seconds. It will be utilized when viewing the results.



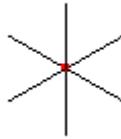
Step 2

Click the Z arrowhead  to view the XY plane parallel to the screen.

Mesh tools → **Create** → **Node...**

X 0
Y 0
Z 0

The node appears as a red dot at the origin.



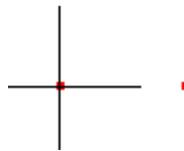
If you don't see the node make sure you've activated the node select mode 

Add more nodes using the following coordinates with units of m:

(0.1,0,0)
(0.1,0.2,0)
(0,0.2,0)



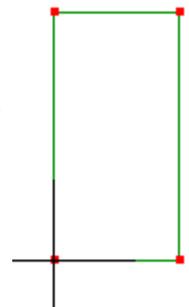
Use the **Fit to window**  to display the nodes.



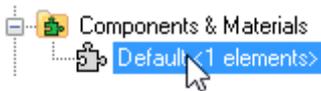
Step 3

Mesh tools → **Create** → **Element...** Select **quad4 shell** and click the four nodes.

The order of nodes will affect the way the element gets subdivided in a following step. So, to keep the order the same, start at the lower left corner and go counter-clockwise.



Step 4



Right click, select **Assign new material**

Geometric tab

Shell/membrane select
Thickness 1 m

Density tab

Density 2707 kg/m³

Thermal tab

Isotropic select

Thermal conductivity 204 W/m/K

Specific heat 896 J/kg/K

Step 5

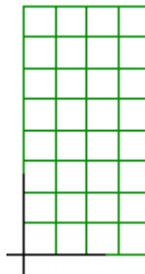
Activate select faces 



Select the element.

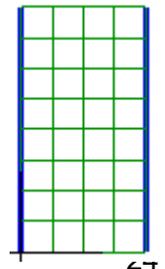
Mesh tools → Refine → Custom...

R 4
S 8



Step 6

Convective heat transfer takes place along the surface of the entire plate. For the FE model, this will be the left and right edges. Click and drag the mouse over the left edge so that the edge faces become selected. Hold the **Ctrl** key down and repeat it for the right edge.



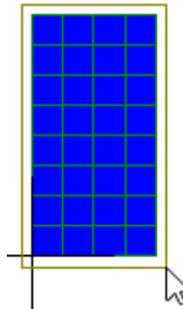
Right click  **Loads & Constraints** then select **New convection**

Ambient temperature 50 °C. Note that the unit is not K.
Heat transfer coefficient 80 W/m²/K



Step 7

Drag a rectangle to select the entire mesh.



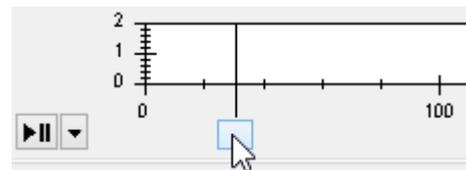
 **Initial Conditions** Right click and select **New temperature**. Ensure °C is selected then type 250 in the text-box.

Step 8

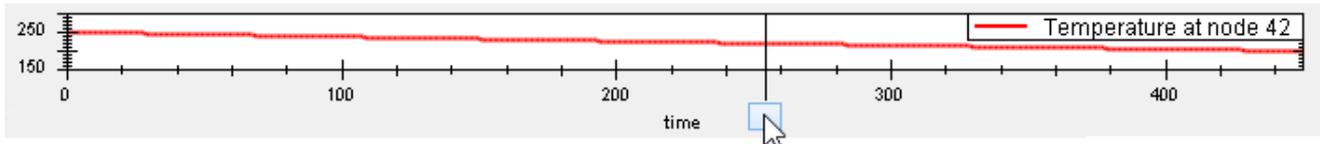
Click  to solve the model. The results are listed in the outline tree below **Solution**.

Click  **Temperature**

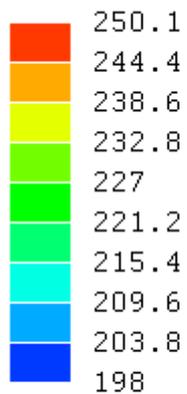
Drag the slider to view the temperature changes with time.



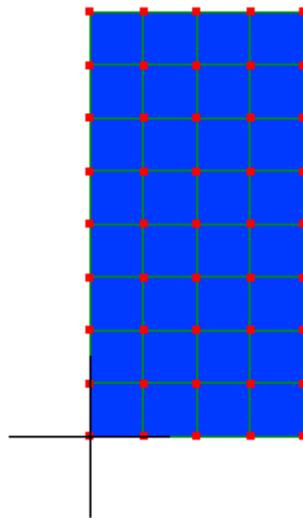
If you click a node the temperature profile of that node will be displayed in the timeline.



Temperature



Time = 440
= 7.333333 × 60



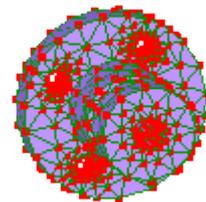
From the results we see that it takes about 440 seconds to reach the temperature of 200°C.

For transient thermal models if your results show a strange oscillation of temperatures every other time-step, use a smaller time-step value and refine the mesh further.

2.3 Heat Flow Through a Solid Part

Step 1

Open **TransientThermalTutorial.liml** from the tutorials folder where Mecway has been installed.



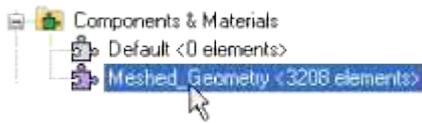
Step 2



Double click **Analysis**
General tab
Time period
Time step

300 s
10 s

Step 3



Right click **Meshed_Geometry** and select **Assign new material**

Density tab

Density 2700 kg/m³

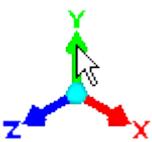
Thermal tab

Isotropic

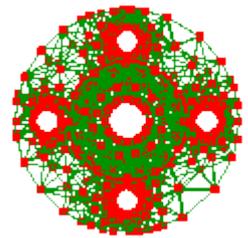
Thermal conductivity 200 W/m/K

Specific heat 890 J/kg/K

Step 4

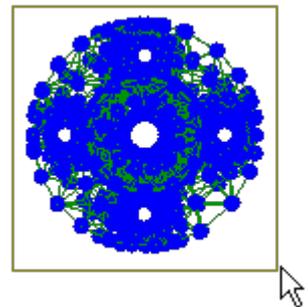


Right click the arrowhead of the Y-axis to display the model parallel to the screen.



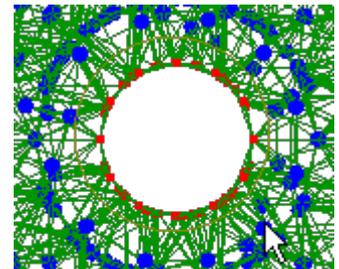
Activate the **Select nodes** mode.

Drag to select the entire mesh.



Edit → Circle selection

Hold the **Ctrl** key down and drag to deselect the nodes of the inner diameter.



Initial Conditions

Right click **Initial Conditions** and select **New temperature**

Apply to <848 Selected nodes>

22 °C

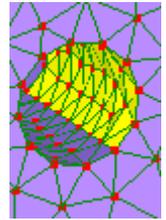
This sets the selected nodes to an initial temperature of 22 °C. The reason the nodes of the inner diameter were deselected is because a constant temperature is going to be applied to the inner diameter. It's not physically possible for a node to be both at an initial temperature of 22 °C and a different constant temperature at the same time.

Step 5

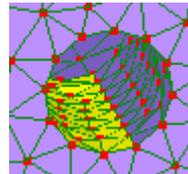
We will now fix the temperature at the internal diameter to 120 for the entire analysis.



Right click **Surface11** and select **New loads & constraints**
→ **New temperature**
Apply to Formula Surface 11
120 °C



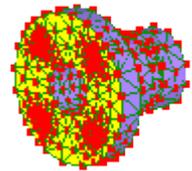
Repeat for **Surface12**



Step 6



Right click **Surface7**
New loads & constraints → **New Convection**
Apply to Ambient temperature Surface 7
22 °C
Heat transfer coefficient 40 W/m²/K



Step 7

 **Solve**

Select any of the results listed below solution.



Drag the slider on the timeline to display the results at the various time steps. Select a node to display a graph of its temperature.

2.4 Thermal Stress

Step 1

Start a new model. **File** → **New**

Double click **Analysis** in the outline tree and select **Thermal Steady State** then press **OK**.



Step 2

Create a new hex8 element by clicking the Quick cube tool-button.

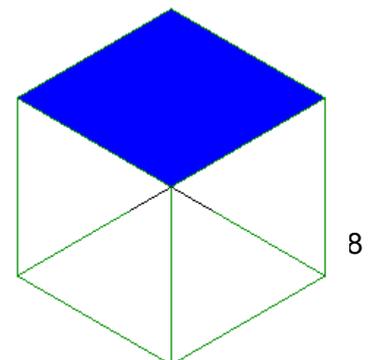
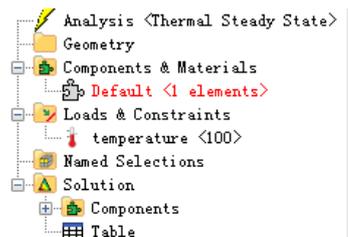


Step 3

Activate select faces 

Click anywhere in the graphics area to clear the selection.

Right click any face and select **Loads &**



8

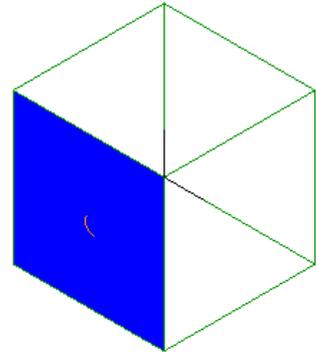
constraints → **New temperature**

Enter the value 100. This constrains the temperature of that face to 100°C.

Step 4

Right click another face and select **Loads & constraints** → **New convection**

Ambient temperature 20
Heat transfer coefficient 400



Step 5

Right click the component **Default <1 elements>** in the outline tree and choose **Assign new material**.

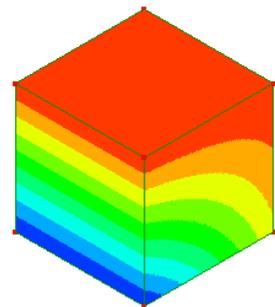
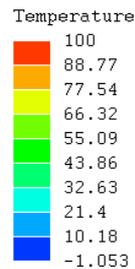
Select **Isotropic** and enter a **Thermal conductivity** of 40. This is 40W/(m.K).

Step 6

 **Solve.**

Click **Temperature** under the **Solution** branch of the outline tree.

Notice that some nodes have an unrealistic temperature lower than the 20°C ambient. This is because of the extremely coarse mesh. In a practical problem, you would refine the mesh to improve accuracy.



Step 7

Double click **Analysis**. Select **Static 3D** 

Step 8

 **Loads & Constraints** Right click **Loads & constraints** and select **Transfer temperatures from solution**

Step 9

 **Loads & Constraints** Right click **Loads & constraints** and select **New thermal stress**
Reference temperature 20

The reference temperature is the initial temperature of the nodes that are now at the temperatures which were transferred from the thermal analysis solution.

Step 10

Right click **Material** and select **Edit**



Mechanical tab

Isotropic

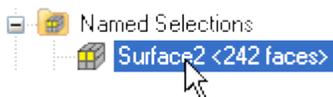
Young's modulus 200E09

Thermal expansion coefficient 11E-06

Density tab

Density 7800

Step 11



Right click any face and choose **Loads & constraints** → **New fixed support**

Step 12



Solve

The solution now shows the cube deformed by thermal expansion.

2.5 Free Vibration of a Cantilever Beam

A cantilever beam of length 1.2m, cross-section 0.2m × 0.05m, Young's modulus 200×10^9 Pa, Poisson ratio 0.3 and density 7860 kg/m^3 . The lowest natural frequency of this beam is required to be determined.

For thin beams, the following analytical equation is used to calculate the first natural frequency :

$$f = (3.52/2\pi)[(k / 3 \times M)]^{1/2}$$

f = frequency

M = mass

$$M = \text{density} \times \text{volume}$$

$$M = 7860 \times 1.2 \times 0.05 \times 0.2$$

$$M = 94.32 \text{ kg}$$

k = spring stiffness

$$k = 3 \times E \times I / L^3$$

I = moment of inertia of the cross-section.

E = Young's modulus

L = beam length

$$I = (1/12)(bh^3)$$

$$I = (1/12) (0.2 \times 0.05^3)$$

$$I = 2.083 \times 10^{-6} \text{ m}^4$$

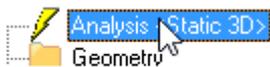
$$k = (3 \times 200 \times 10^9 \times 2.083 \times 10^{-6}) / 1.2^3$$

$$k = 723.379 \times 10^3 \text{ N/m}$$

$$f = (3.52/2 \times 3.14) [(723.379 \times 10^3 / 3 \times 94.32)]^{1/2}$$

$$f = 28.32 \text{ Hz}$$

Step 1



Right click, **Edit**, then select **Modal Vibration 2D**.

Number of modes 3

Step 2

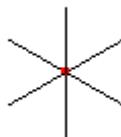
Mesh tools → **Create** → **Node...**

X 0

Y 0

Z 0

The node appears as a red dot at the origin.



If you don't see the node make sure you've in the node select mode 

Click the Z arrowhead  to view the XY plane parallel to the screen.

Add more nodes using the following coordinates, all with units of m.

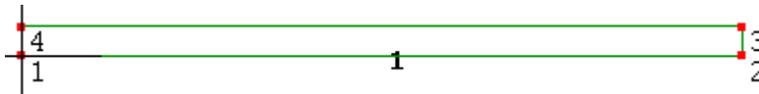
- (1.2,0,0)
- (1.2,0.05,0)
- (0,0.05,0)



Use the **Fit to window**  to display the nodes.

Step 3

Mesh tools → **Create** → **Element...** Select **quad4 plane** and click the four nodes. The order of the clicked nodes will affect the orientation of the mesh refinement that will be done in the next step. In this tutorial the element is formed using the node order 1,2,3,4.



Step 4



Right click, **Assign new material**

Geometric tab select **Shell/membrane**
Thickness 0.2 m

Mechanical tab
Isotropic select
Young's modulus 200 GPa
Poisson's ratio 0.3

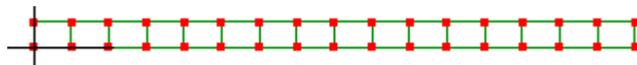
Density tab
Density 7860 kg/m³

Step 5

Mesh tools → **Refine** → **Custom...**

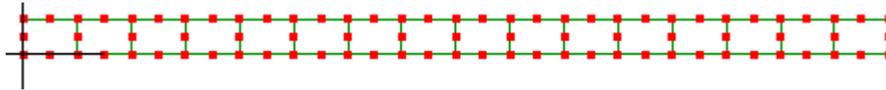
Number of subdivisions

- R** 16
- S** 1



Step 6

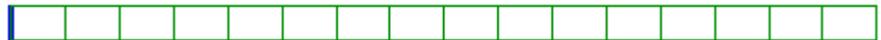
Elements with mid-side nodes (quadratic) are more accurate in bending problems. The four node quadrilaterals will be changed into eight node quadrilaterals using **Mesh tools** → **Change element shape...**, select **quad8**



Step 7

Activate select faces 

Click to select the left edge face.



Right click  **Loads & Constraints** then select **New fixed support**

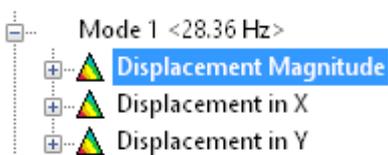


Step 8

Click  to solve the model.

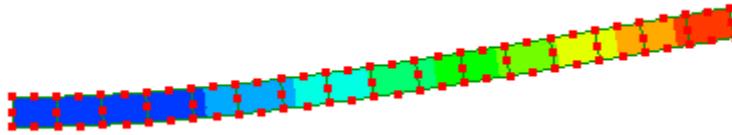
The results are listed in the outline tree below **Solution**. Click  **Mode 1 <28.37391>**

Click the animation  tool-button to view the mode shape. For this problem, the lowest mode shape is expected to vibrate back and forth in bending.



The displacement values of a modal analysis are only meaningful relative to each other. The magnitude of the displacement of a real structure depends on load history and damping which are not

modeled by modal vibration analysis.

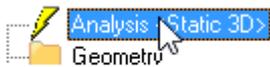


The frequency for the lowest mode matches the hand calculation result of 28.32 Hz

2.6 Dynamic Response of a Crane Frame

A dynamic response analysis is difficult to estimate values for by simple hand calculations. This tutorial will simply illustrate the dynamic response analysis work-flow of a crane's truss framework.

Step 1



Right click, **Edit**. Select **Dynamic Response 2D**
Time period 0.5 s
Time step 0.005 s

Step 2

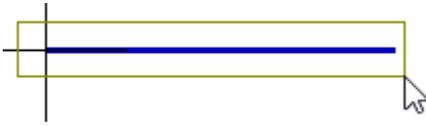
Mesh tools → **Create** → **Curve generator...** select **Straight line**
X1 0
Y1 0
Z1 0
X2 20, unit m
Y2 0
Z2 0
Number of nodes 5

Click OK twice then click the Z arrowhead  to view the XY plane parallel to the screen.

Use the **Fit to window**  to display the whole mesh.



Activate select elements 



Drag to select all the elements. Right click on the selected elements and select **Element properties**. Select the check-box next to **Truss** to convert these elements from beam elements to truss.

Step 3

With all elements still selected

Mesh tools → **Move/copy...**

Y 5, unit m

Copy select

Click **Apply** then **Close**.

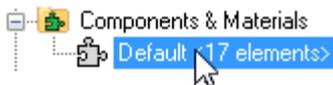


Step 4

Mesh tools → **Create** → **Element...** Select **line2** and click the nodes to form the pattern shown.



Step 5



Right click, **Assign new material**

Geometric tab

General section

Cross-sectional area select 0.0225, unit m²

Mechanical tab

Isotropic

Young's modulus select 20E09, unit Pa

Poisson's ratio 0.3

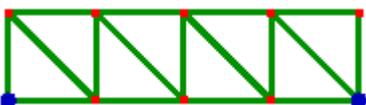
Density tab

Density 7860, unit kg/m³

Click **OK**.

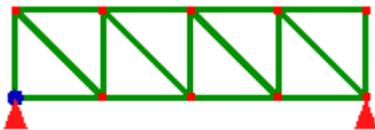
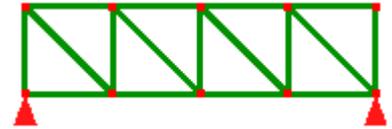
Step 6

Activate select nodes 



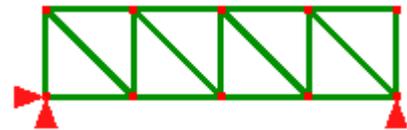
Select these two nodes. Hold the **Ctrl** key down while selecting the second node so as not to deselect the first node.

Right click  **Loads & Constraints** then select **New displacement**, select **Y** and click **OK**.

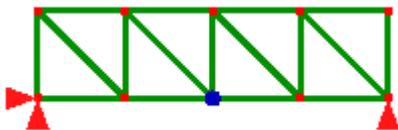


Select this node alone.

Right click  **Loads & Constraints** then select **New displacement**, select **X** and click **OK**.



Step 7



Select this node.

Right click  **Loads & Constraints** then select **New force**
Y

Table	select
	0.09 0
	0.1 -1

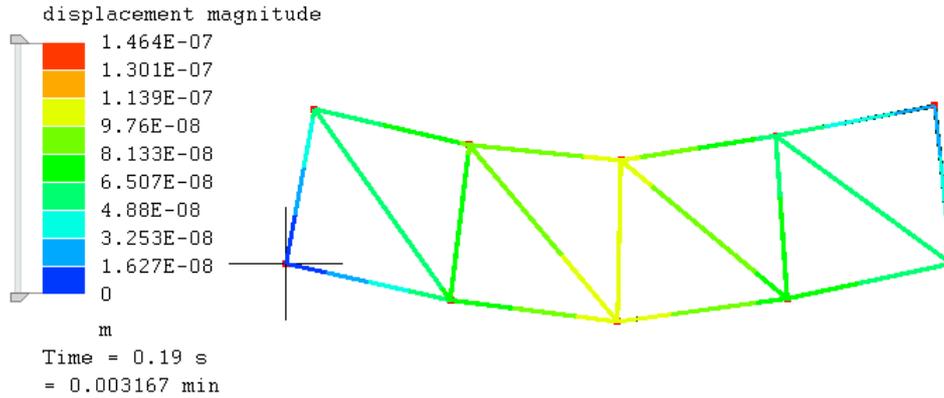
Choose units **s** and **N** then click **OK**.

Step 10

Click  to solve the model.

The results are listed in the outline tree below **Solution**. Click one of them to display it.

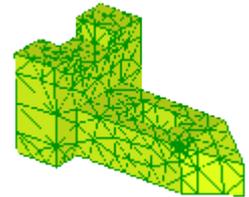
Click the play/pause button in the timeline to view the dynamic response of the structure. Click **Solution** → **Deformed view** to deform the structure while it's animated.



2.7 Dynamic Response of a Solid Part

Step 1

Open **DynamicResponseTutorial.liml** from the tutorials folder where Mecway has been installed.

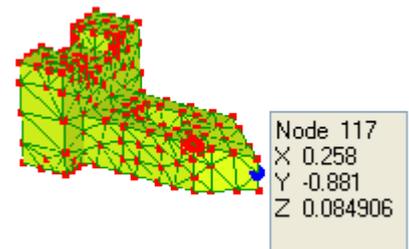


Step 2

Analysis <Dynamic Response 3D> Double click **Analysis**
 Geometry Time period 0.003 s
 Time step 0.0004 s

Step 3

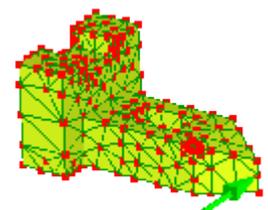
Activate the **Select nodes** mode and select this node.



An applied force will ramp up linearly from 0 to a maximum 500 N then down to 0. At each time step Mecway will determine the force by interpolating between the specified values.

Loads & Constraints Right click **Loads & Constraints** and select **New force**
Apply to <Selected nodes>
X
Table

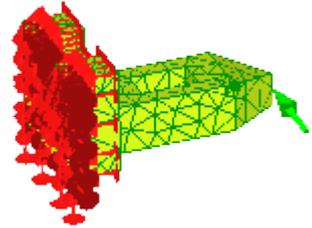
0	0
0.0015	500
0.003	0



Step 4



Right click Surface15 and select **New loads & constraints** → **New fixed support**

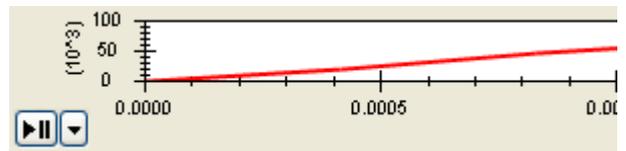


Step 5

Solve

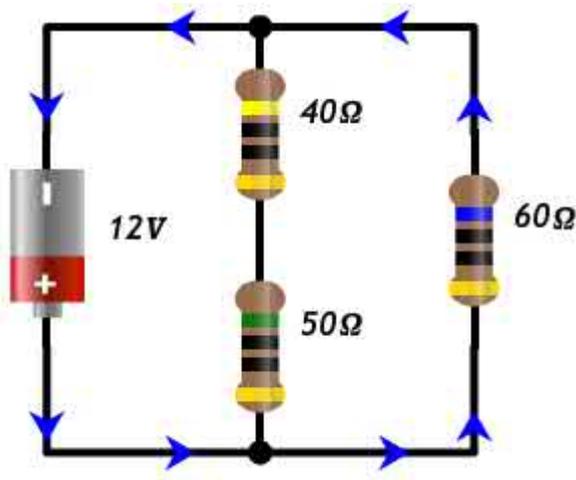
Select any of the results below the Solution group in the outline tree.

Drag the slider on the timeline to display the results at the various time steps. Select a node to display a graph of its displacement.



2.8 DC Circuit

The following circuit will be solved for the voltage drop across the resistances, and the currents.



Current flowing through the 40Ω and 50Ω resistors is

$$12V / (40\Omega + 50\Omega) = 0.13A$$

Current flow through the 60Ω resistor is

$$12V / 60\Omega = 0.2A$$

The voltage drop across the 40Ω resistor is

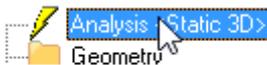
$$0.13A \times 40\Omega = 5.3V$$

Voltage drop across the 50Ω resistor is

$$0.13A \times 50\Omega = 6.7V$$

Voltage drop across the 60Ω resistor is 12V

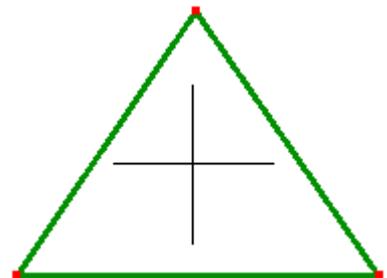
Step 1

 Right click, **Edit**. Select **DC Current Flow**.

Step 2

Click the Z arrowhead  to view the XY plane parallel to the screen.

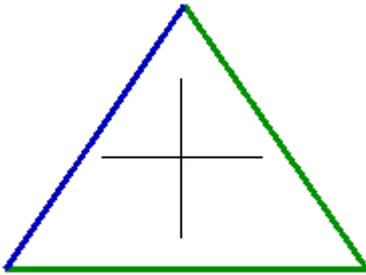
Click the **New element** tool  and click in the graphics area to form the following three line elements. For the 2nd and 3rd elements, be careful to click on existing nodes so they are linked together.



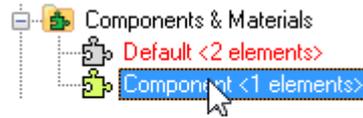
Step 3

As each element has a different resistance, they need to be separate components.

Activate select elements 



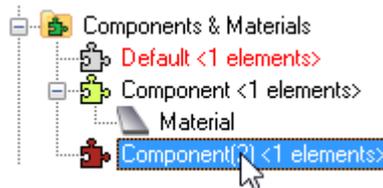
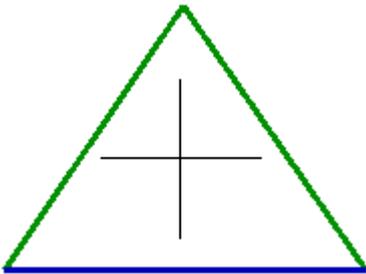
Right click on the selected element and select **Add elements to new component**



Right click on the new component and select **Assign new material**.

Electric tab
Resistor select
Resistance 40 Ω

Repeat this step for the second element.



Electric tab
Resistor select
Resistance 50 Ω

As there is only one element left in the **Default** under **Components & Materials**, right click to **Assign a new material**.

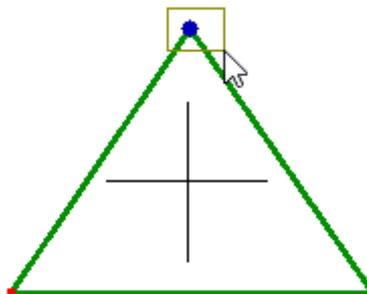


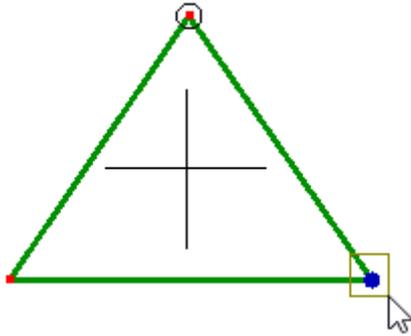
Electric tab
Resistor select
Resistance 60

Step 4

Activate **Select nodes** 

Right click on this selected node then select **Loads & constraints** → **New electric potential** type 0 in the text-box.





Right click on this selected node then select **Loads & constraints** → **New electric potential**. Select units **V** then type 12 in the text-box.

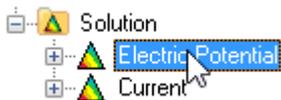
A 12V voltage has now been applied to the circuit.

Step 5

Click  to solve the model.

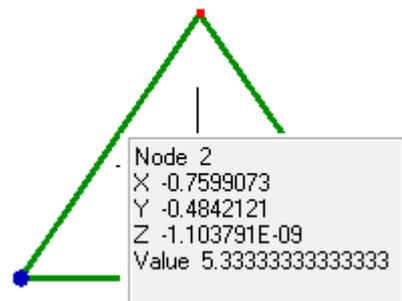
Electric potential results are listed in the outline tree. To add current, right click **Solution** and choose **New electric** → **current**.

The results are listed in the outline tree below **Solution**.



Select one of the field variables and then click a node for a read-out of that value.

The sign of a current indicates its direction. If it is -ve, the current is flowing in the direction in which you formed the element going from the first node of that element to the second node. If the value is +ve, current is flowing in the opposite direction from the second node to the first.



For the currents, read the element values instead of the node averaged value. Nodal averaging can be misleading, for example the -0.2A in the 60Ω resistor averaged with the 0.1333A in the 40ohm resistor gives a non-existent 0.03333A.

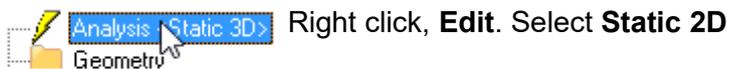
2.9 Electrostatic Analysis of a Capacitor

The electric field in a simple two plate capacitor will be modeled. The plates are 0.005 m apart with air in between, and a potential difference of 1.5V.

The expected electric field, $E = \text{potential difference} / \text{distance between plates}$

$$E = 1.5 / 0.005 = 300 \text{ V/m}$$

Step 1



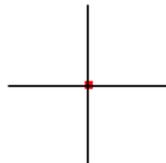
Step 2

Click the Z arrowhead  to view the XY plane parallel to the screen.

Mesh tools → **Create** → **Node...**

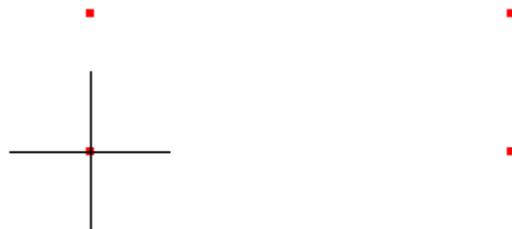
X 0
Y 0
Z 0

A red dot appears at the origin.



Add more nodes using the following coordinates, all with units of m.

(0.015,0,0)
(0.015,0.005,0)
(0,0.005,0)

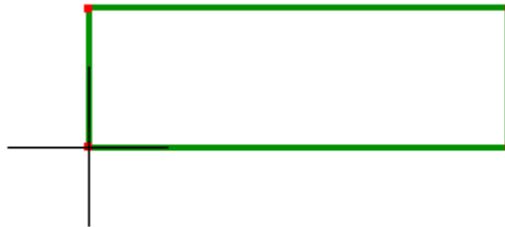


Use the **Fit to window**  to display the nodes

Step 3

In this tutorial the area between the two plates of the capacitor will be defined using two node line elements, and then the 2D automesh will then be used to fill the region with quadrilateral or triangle elements.

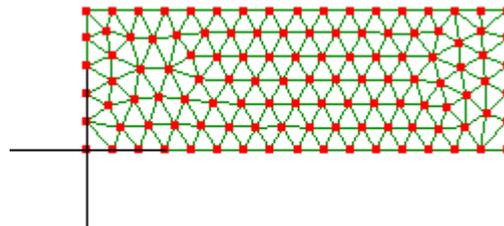
Mesh tools → **Create** → **Element...** Select **line2** and click the four nodes to form the area between the two plates.



Step 4

Mesh tools → **Automesh 2D...**

Maximum element size 0.001 m
accept the rest of the defaults



Step 5



Right click, **Assign new material**

Geometric tab

Thickness 1 m

Electric tab

Isotropic select

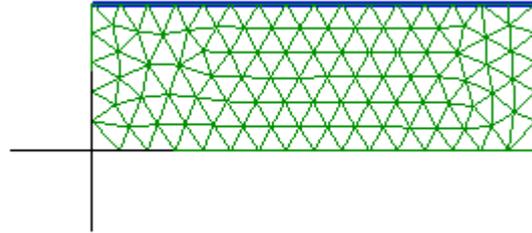
Relative permittivity 1

Step 6

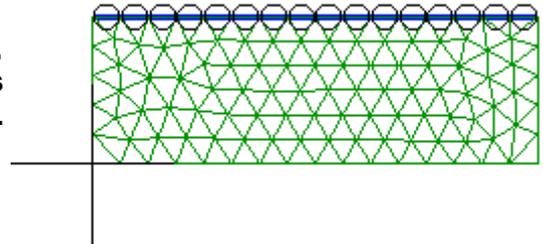
Activate **Select faces** 

Drag to select the faces along the top edge.

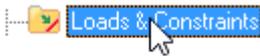
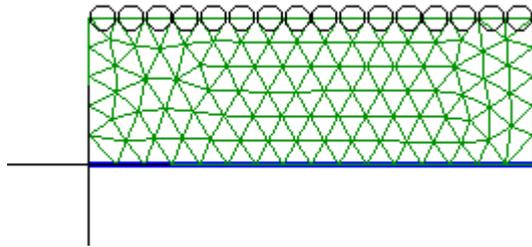
The element faces will become selected. Although they appear to be edges, in reality they are faces normal to the screen because of the thickness dimension that is not displayed.



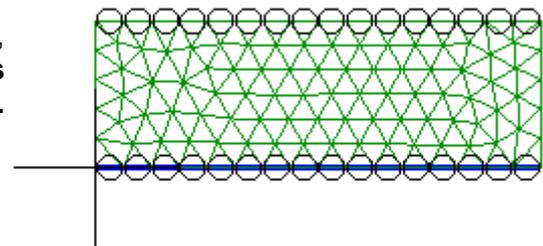
While the faces (edges) are selected, right click on **Loads & Constraints** then click **New electric potential**. Select V and type 1.5 in the text-box.



Drag to select the faces at the bottom.



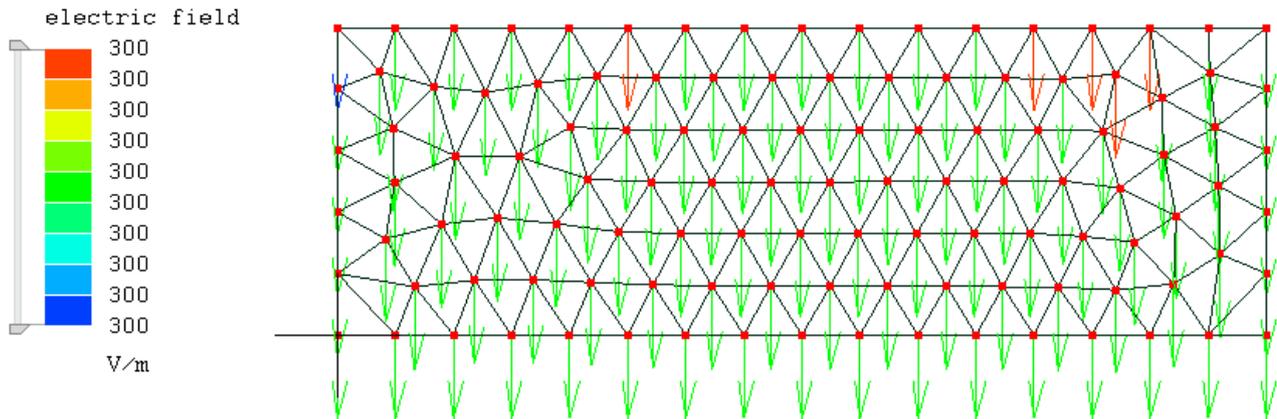
While the edges (faces) are selected, right click on **Loads & Constraints** then click **New electric potential**. Accept the default 0 value.



Step 7

Click  to solve the model.

Some results are listed in the outline tree. Right click **Solution** and choose **New electric** → **electric field**. Then click the new **electric field** branch of the outline tree to display the field. It matches the expected value of 300 V/m.



2.10 Acoustic Analysis of an Organ Pipe

We will predict the musical note produced by an 8 ft long 'open wood' organ pipe. An actual pipe would be rectangular in section, but here we make only a 2D model. The pipe will be 120 mm wide with a 30 mm mouth at one end where air is blown in, and an opening at the other end for the air to exit. The wood is 10 mm thick.

In acoustic analysis the air within the pipe needs to be meshed and not the pipe itself. The mesh needs to include a large volume of the air in the room outside the pipe which is connected to the mouth and the open end. This scheme allows Mecway to model leakages and openings. We are interested only in the resonant modes of the organ pipe, but Mecway will also predict resonances in the air outside. An important part of this case study, therefore, is to show how some understanding of the physics is required in order to interpret the results and identify the appropriate modes.

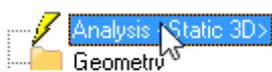
A rough calculation will be made for the expected frequency.

An end correction factor must be added to the length of the pipe. This correction factor is equal to the width of the pipe. So the effective length of the pipe is $2.4384 \text{ m} + 0.12 \text{ m} = 2.5584 \text{ m}$

When $1/2$ a wavelength fits into the pipe it will be of wavelength $\lambda = 2 \times 2.5584 \text{ m} = 5.1168 \text{ m}$

The speed of sound in air at 15°C is 340 m/s , so the expected frequency will be $f = v/\lambda = 340/5.1168 = 66.45 \text{ Hz}$.

Step 1



Double click **Analysis**, then select **Acoustic Resonance 2D**
Number of modes 20

To ascertain the fundamental pitch we need to examine the results for a number of pitches. We will also look at harmonics.

Step 2

Click **Quick square** to create a quad4 element.

Click the Z arrowhead  to orient the XY plane parallel to the screen.

Step 3

Drag to select the two right-most nodes then use **Mesh tools** → **Node coordinates** and change their X coordinates:

X 10 m

Y (blank)

Z 0

Use **Edit** → **Select nodes by formula** and enter

0 < y

then click **Apply** and **Close** to select the two top-most nodes. Use **Mesh tools** → **Node coordinates** again to change their Y coordinates:

X (blank)

Y 2 m

Z 0

Press **Fit to window**  so you can still see the whole mesh.



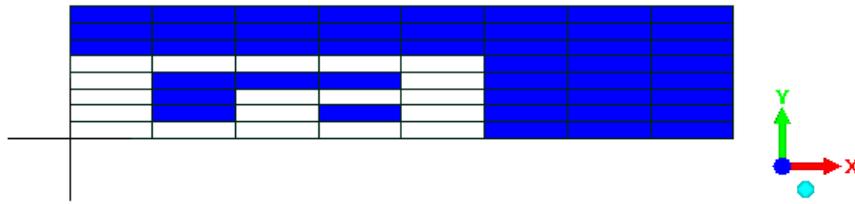
Step 4

Click a blank place in the graphics area to clear the selection.

Click **Refine x2**  three times.

Step 5

Change to Select elements mode  and select the elements shown then delete them by pressing the **del** key. To select the large regions, drag the mouse. Hold the **ctrl** key to add elements to the selection.



Step 6

Use **Mesh tools** → **Move/copy...** and enter an **X** displacement of **-2.5 m**.

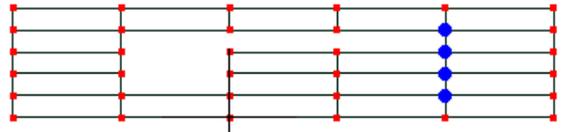
Press **Fit to window**  so you can still see the whole mesh.

Step 7

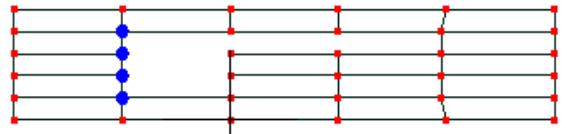
Here we'll set the length of the pipe and width of the mouth.

Change to **Select nodes** mode 

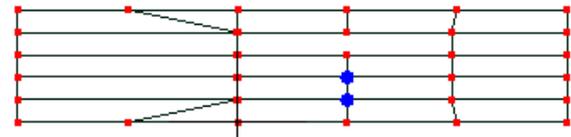
Select the 4 nodes shown then right click one of them and choose **Node coordinates**. Specify **X** as **8 ft** and press **OK**.



Select the 4 nodes shown then right click one of them and choose **Node coordinates**. Specify **X** as **-10 mm** and press **OK**.



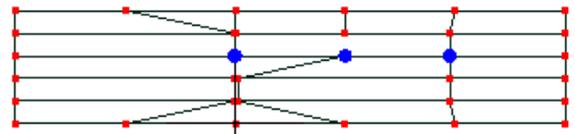
Select the 2 nodes shown then right click one of them and choose **Node coordinates**. Specify **X** as **30 mm** and press **OK**.



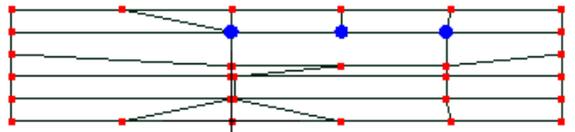
Step 8

Here we'll set the pipe's width and wall thickness.

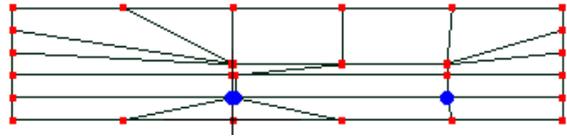
Select the 4 nodes shown then right click one of them and choose **Node coordinates**. Specify **Y** as **620 mm** and press **OK**.



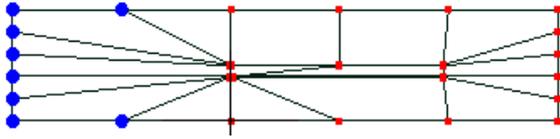
Select the 4 nodes shown then right click one of them and choose **Node coordinates**. Specify **Y** as **630 mm** and press **OK**.



Select the 4 nodes shown then right click one of them and choose **Node coordinates**. Specify **Y** as **490 mm** and press **OK**.



Step 9



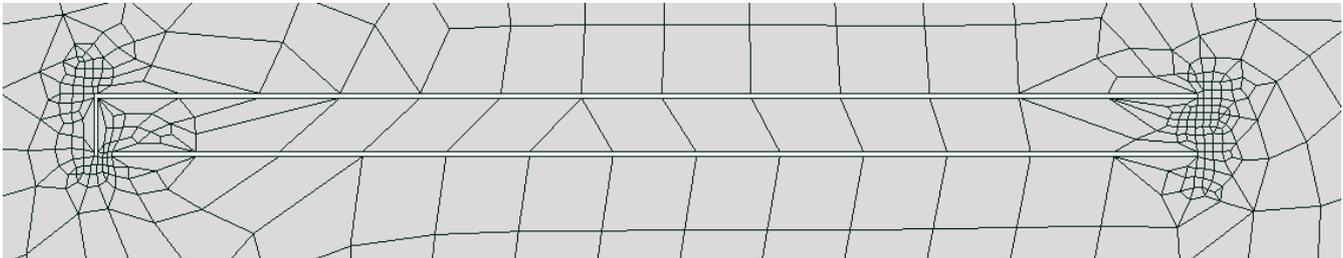
Select these 8 room air nodes to the left of the pipe and use **Mesh tools** → **Scale...** with an **X** factor of **0.5**. Then click a blank space in the graphics area to clear the selection.

Step 10

Generate a finer mesh using **Mesh tools** → **Automesh 2D...**

Max. element size: 200 mm

Quad dominant: checked



Step 11



Right click the **Default** component and select **Assign new material**

Mechanical tab

Isotropic

select

Speed of sound

340

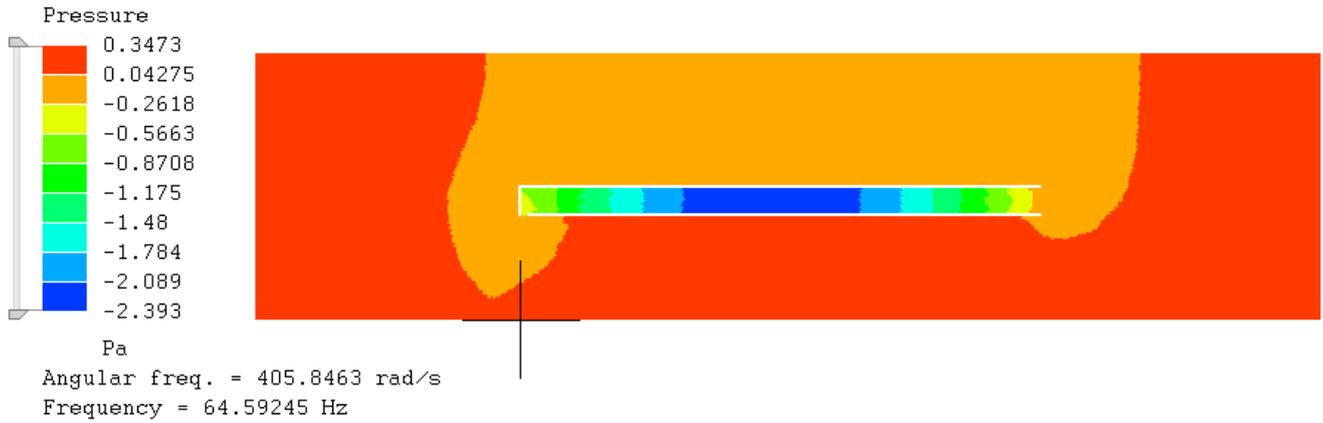
Step 12

The mesh building is now complete so click  to solve the model.

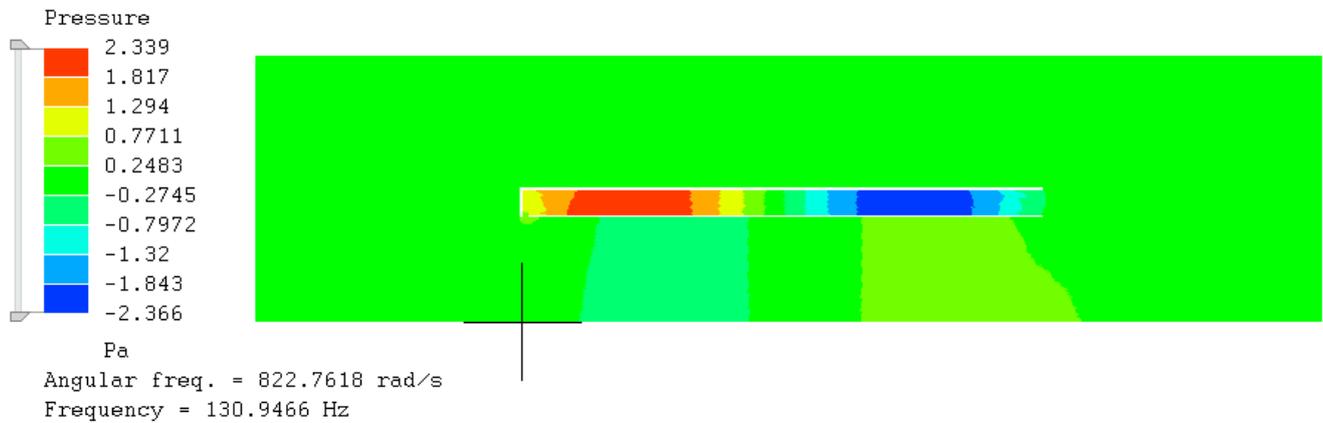
The modal frequencies are listed in the outline tree below Solution. Click the nodes for a read-out of the field variable there.

The subtlety in this case study is in interpreting the results. From the 20 calculated modes we want to pick out the fundamental pitch of the pipe itself. Click through the various modes and observe that most show red, yellow and blue patches for the room air elements.

The 4th mode at 64.59 Hz is different because the 'room' is almost at constant pressure (color). Zoom in on the pipe and notice that the magnitude of the maximum pressure inside the pipe is much higher than anywhere outside the pipe. This identifies mode 4 as the fundamental pitch of the organ pipe. Half a wavelength fits inside the pipe. The frequency matches the hand calculated value very well.



Similarly Mode 8 at 131 Hz has an almost constant pressure in the room and high, variable pressure in the pipe. This is the first harmonic at twice the frequency of mode 4, sounding one octave higher. A full wavelength fits inside the pipe.



2.11 Buckling of a Column

The eigenvalue buckling of a column with a fixed end will be solved. The column has a length of 100mm, a square cross-section of 10mm and Young's modulus 200000 N/mm².

The critical load for a fixed end Euler column is $\pi^2 EI / (4L^2)$

E = Young's modulus

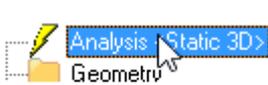
I = moment of inertia

$$I = 10^4 / 12 = 833.33 \text{ mm}^4$$

L = length

$$\text{Critical load} = \pi^2 200000 \times 833.33 / (4 \times 100^2) = 41123.19$$

Step 1



Right click, **Edit**. Select **Buckling 2D Beam**.

Number of modes 3

Shift point 1

Step 2

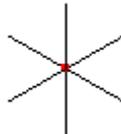
Mesh tools → **Create** → **Node...**

X 0

Y 0

Z 0

The node appears as a red dot at the origin.



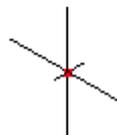
If you don't see the node, make sure you've activated the node select mode 

Add another node using the following co-ordinates.

(0, 100 mm, 0)



Use the **Fit to window**  to display the nodes.



Step 3

Mesh tools → **Create** → **Element..**
nodes.



Select and click the two



Step 4



Right click, select **Assign new material**

Geometric tab

General section

2nd moment of area about W

Cross sectional area

select

833.33 mm⁴

100 mm²

Mechanical tab

Isotropic

Young's modulus

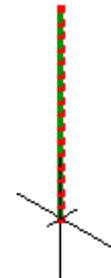
select

200000 MPa

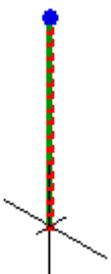
Step 5

Mesh tools → **Refine** → **x2**  or four times.

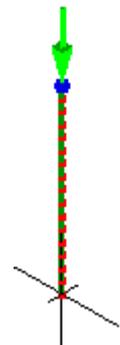
To see the mesh refinement switch to the node select mode 



Step 6



Select this node then right click  **Loads & Constraints** and select **New force**. In the **Y** box, select unit **N** and type -1.



Step 7

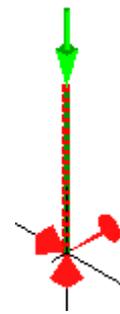
Activate select nodes 



Select the node at the bottom then change to face selection mode 

Now the bottom end face is selected.

Right click  **Loads & Constraints** and select **New fixed support**



Step 8

Click  to solve the model.

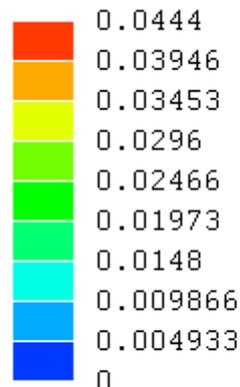
The results are listed in the outline tree below **Solution**.

Click the animation tool-button  to view the buckling mode shape.

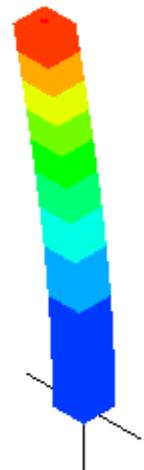
As the load used for the model was 1N, the load at failure is $1 \times 41123.27 = 41123.27$ N, which approximately matches the hand calculated value of 41123.19.

The error in the last two digits can be reduced by increasing the value of **Shift point** in step 1 from 1 to 10000. This higher value is chosen to be slightly less than the lowest buckling factor.

Displacement Magnitude



Buckling factor = 41123.27



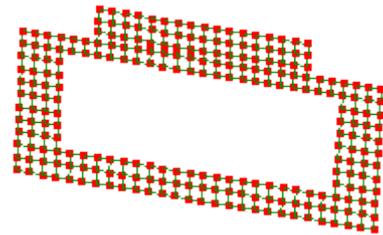
That concludes our overview of the types of analysis which Mecway can carry out. Bear in mind that mechanical analysis can generally be done using orthotropic materials such as wood or carbon fiber composites, as well as the more familiar isotropic materials.

Operations

3.1 Extrude

Step 1

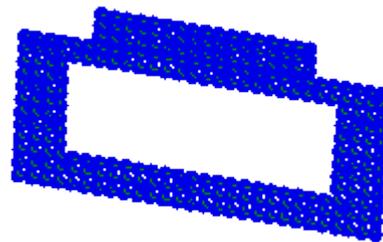
Open **ExtrudeTutorial.liml** from the tutorials folder where Mecway has been installed.



Step 2

Activate **Select faces** 

Drag the mouse over the entire mesh so that it becomes selected.



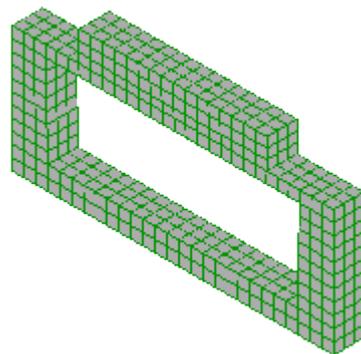
Step 3

Mesh tools → **Extrude...**

Thickness 5 m

Number of subdivisions 3

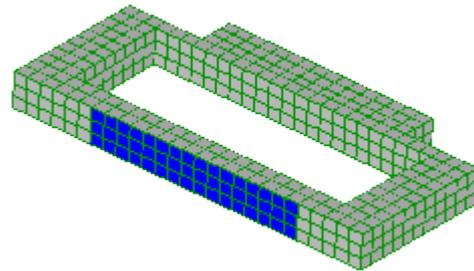
Direction +Z



Step 4

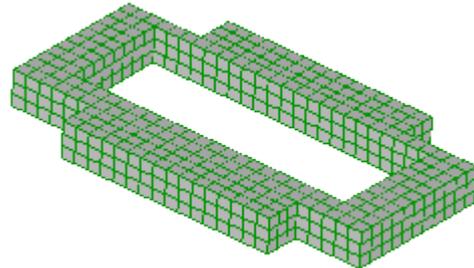
The extrusion step turned off **Select faces** so reactivate it again.

Select the faces at the bottom.



Mesh tools → **Extrude...**

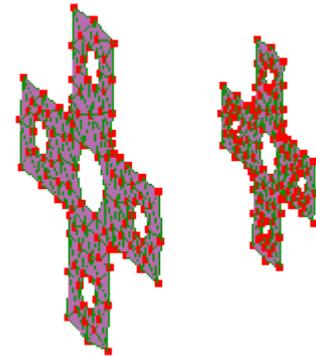
Thickness 5 m
Number of subdivisions 3
Direction +Normal



3.2 Loft

Step 1

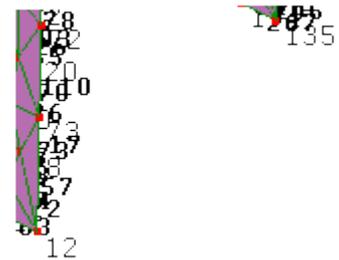
Open **LoftTutorial.liml** from the tutorials folder where Mecway has been installed.



Step 2

Display node and element numbers. 

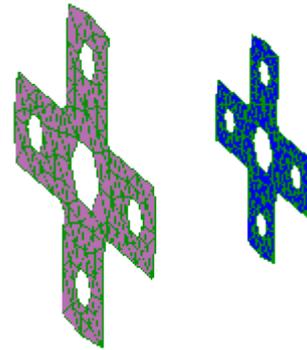
Note down the node numbers of any two corresponding nodes on each profile. In this example the bottom corner node numbers are 12 and 135.



Step 3

Activate **Select faces.** 

Drag to select the profile of node number 135.



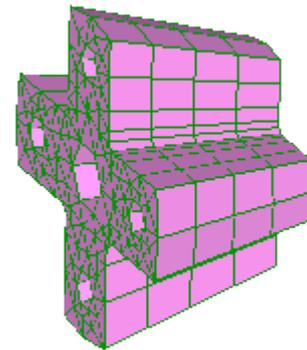
Step 4

Mesh tools → **Loft...**

Number of subdivisions 4

A node in selected faces 135

The corresponding node 12



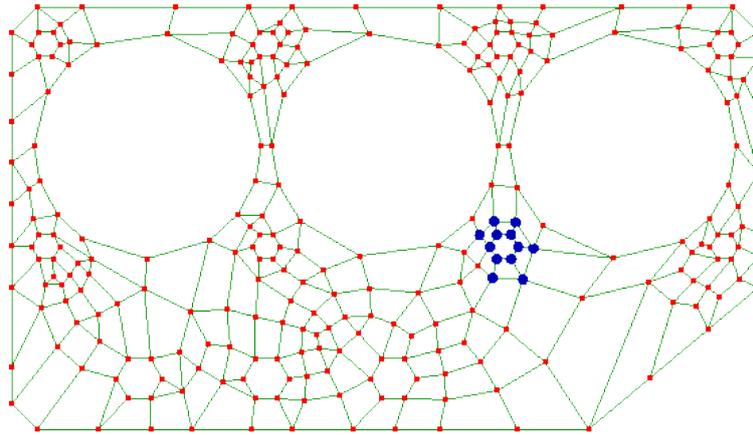
3.3 Refine Local 2D

Step 1

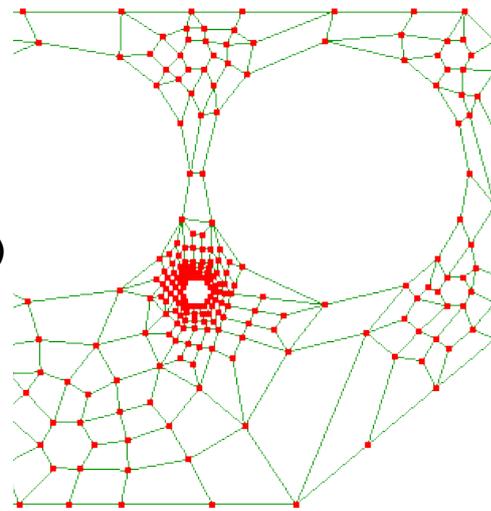
Open **RefineLocal2dTutorial.liml** from the tutorials folder where Mecway has been installed.

Step 2

Drag the mouse to select the nodes shown below. Don't worry if you selected a few extra nodes, this is only an illustration.

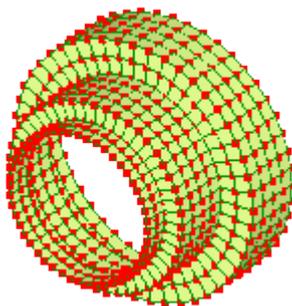


Step 3



Mesh tools → Refine → x3 (1D/2D elements only)

3.4 Mirror Symmetry



Step 1

When the geometry and loads are symmetric, you can model just the part on one side of the symmetry plane(s). This revolved shell can be modeled as only one quadrant. Open **MirrorSymmetryTutorial.liml** from the tutorials folder where Mecway has been installed.



Step 2

The quarter segment's vertical plane of symmetry is the XY plane and its horizontal plane of symmetry is the ZX plane. Its axis of revolution is the X-axis.

Activate **Select nodes** 

Select the nodes lying in the vertical plane of symmetry. Constraints need to be applied to keep these nodes from moving out of the XY plane, while allowing them the freedom to move within the XY plane.

Right click the selected nodes

Loads & constraints → **New displacement** and select the **Z** option.

Since shell nodes have rotational degrees of freedom, the nodes in the XY plane must be prevented from rotating about the X and Y axes.

Right click on the selected nodes again

Loads & constraints → **New node rotation** and select the **X** option.

Repeat for the **Y** axis.

Step 3

While still in **Select nodes**  mode, select the nodes lying in the horizontal plane of symmetry. Constraints need to be applied to keep these nodes from moving out of the ZX plane or rotating about any axis lying in that plane.

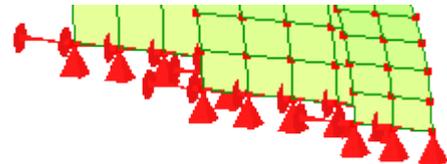
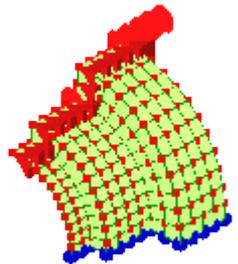
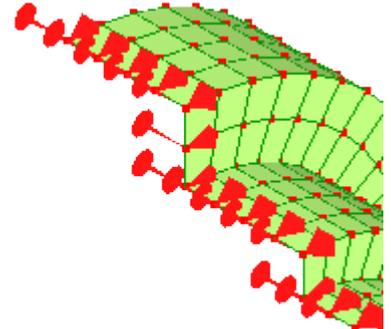
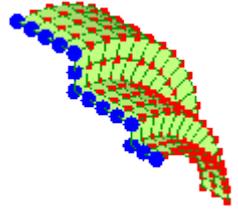
Right click on the selected nodes

Loads & constraints → **New displacement** and select the **Y** option.

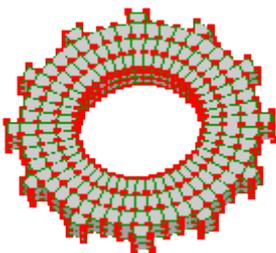
Right click on the selected nodes again

Loads & constraints → **New node rotation** and select the **X** option.

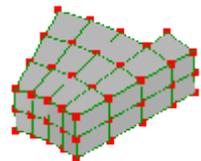
Repeat for the **Z** axis.



3.5 Cyclic Symmetry

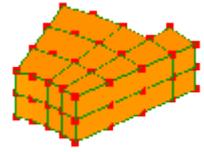


Instead of modeling the entire wheel using solid elements, only one segment will be modeled using cyclic symmetry.



Step 1

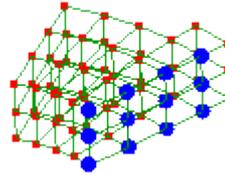
Open **CyclicSymmetryTutorial.liml** from the tutorials folder where Mecway has been installed.



Step 2

Activate **Select nodes** 

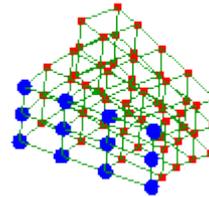
Hold the **Ctrl** key down and select these 12 nodes with the mouse.



Right click on one of the selected nodes and select **Add nodes to new named selection**.

Click a blank part of the graphics area to deselect the nodes

Hold the **Ctrl** key down and select these nodes with the mouse.



Right click on one of the selected nodes and select **Add nodes to new named selection**.

Step 3

Right click Loads & Constraints and select **New cyclic symmetry**.  **Loads & Constraints**

Master nodes	Unnamed
Slave nodes	Unnamed(2)
Axis of symmetry	Y
Number of segments	12

Step 4

Click Solve  then click on some of the modes to see that the solution has expanded the model to a complete ring and that it includes mode shapes which are not cyclically symmetric as well as those that are.

3.6 CAD Work-flow

Step 1

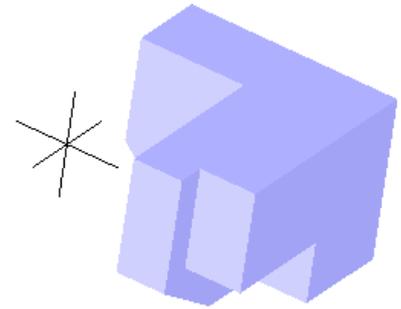
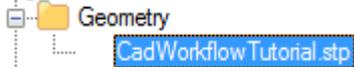
Use **File** → **Open** or right click  **Geometry** and select **Import STEP file**

Open the file **CadWorkflowTutorial.stp** from the tutorials folder where Mecway has been installed.

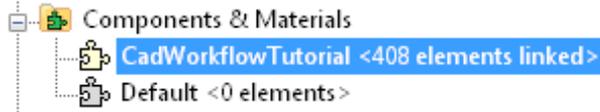
Drag anywhere in the graphics area using the middle mouse button to rotate it so the faces are distinct.

Step 2

Right click the file name and select **Generate Mesh**



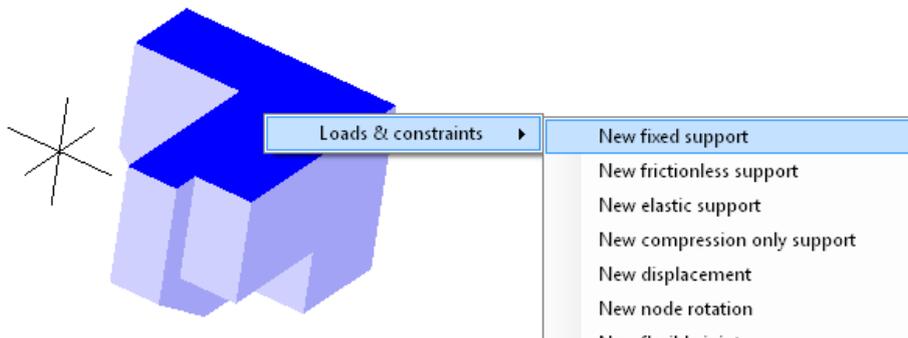
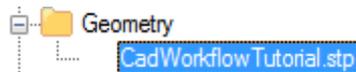
Step 3



Right click the component and select **Assign new material**. In the **Mechanical** tab select **Isotropic** then type 200E09 in the text box for Young's modulus

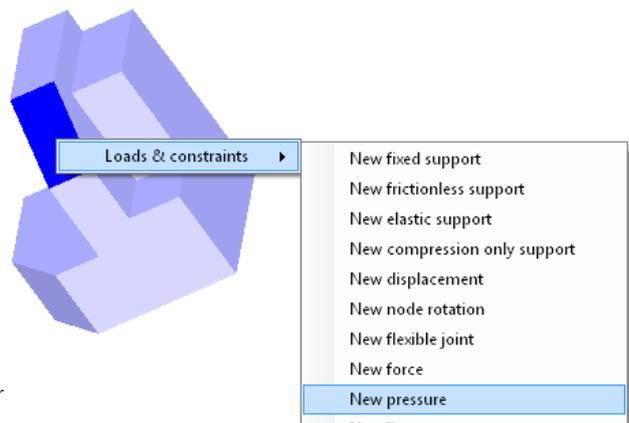
Step 4

Click the file name to return to the geometry view then right click a surface and select **Loads & constraints** then **New fixed support**.



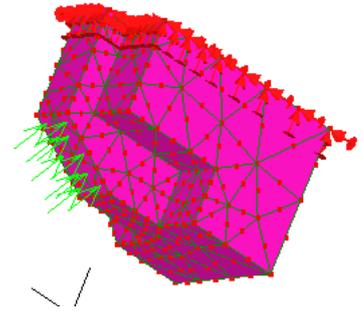
Step 5

Use the middle mouse button to rotate the model, then right click the inclined surface and select **Loads & constraints**, then select **Pressure**. Type **1000** to apply a pressure to the selected surface.



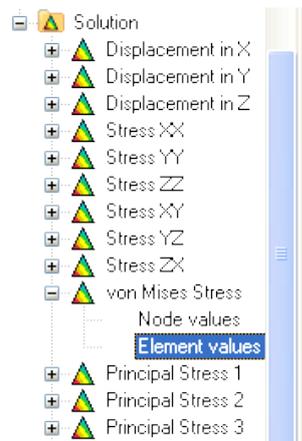
Step 6

Click **Components & Materials** in the outline tree to switch to the mesh view. Click **Show element surfaces**  to give a clearer view.

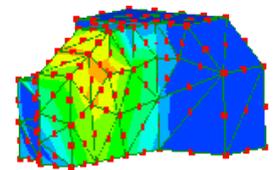
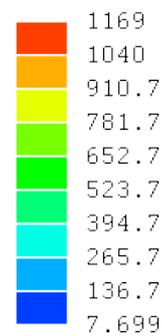


Step 7

Click **Solve**  then view the results



von Mises Stress



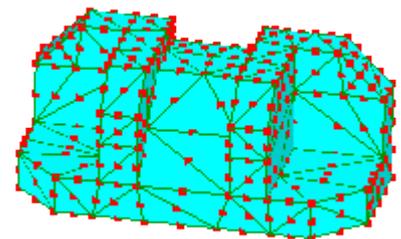
3.7 Local Refinement with CAD Models

Step 1

Open **RefineLocal3dTutorial.stp** from the tutorials folder where Mecway has been installed.

Step 2

Right click the filename and select **Generate mesh**

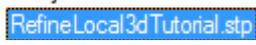


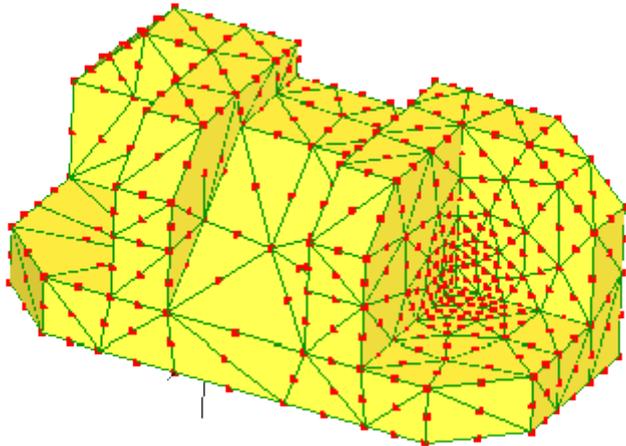
Step 3

Activate **Select nodes** 

Right click a node and choose **New point refinement** then press **OK** to accept the default values.

Step 4

Right click the filename and select **Generate mesh**  Geometry 



Multiple local refinements can be applied following the same procedure.

3.8 Mixed Materials

Step 1

Open a new instance of Mecway.

Step 2

Click **Quick cube** 

Step 3

Click **Refine** 

Step 4

Right click the **Default** component in the outline tree and select **Assign new material**. Choose **Isotropic** and enter:

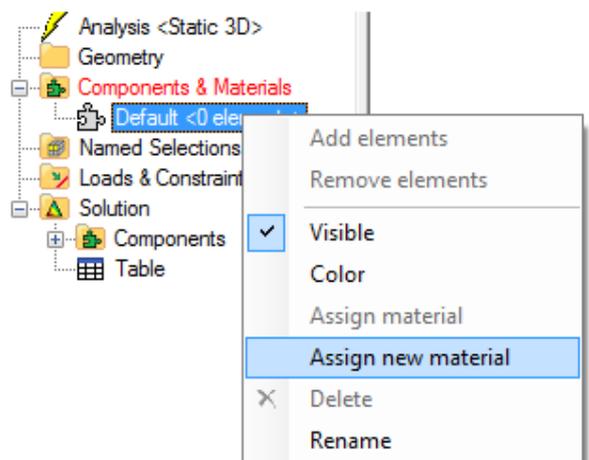
Young's modulus 200e9

Poisson's ratio 0.3

Click **OK**

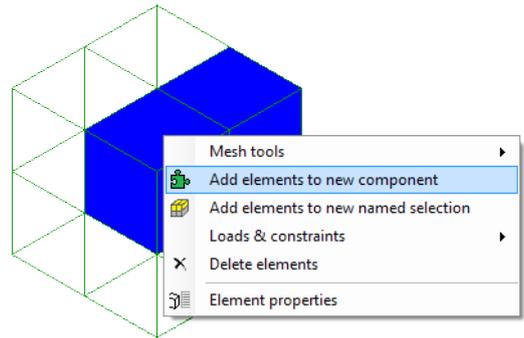
Step 5

Change to **Select elements** mode  and select two of the 8 elements. Hold the **Ctrl** key while clicking on each element.



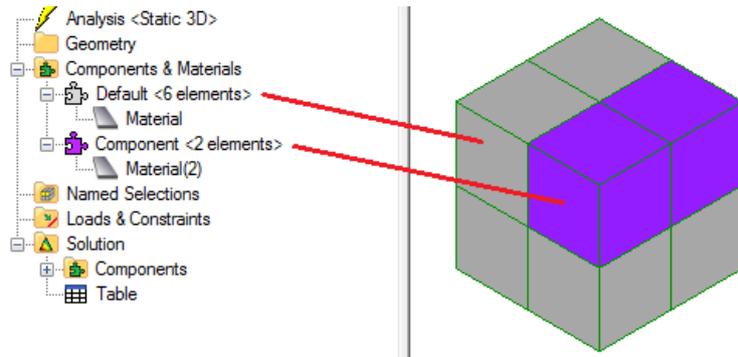
Step 6

Right click one of the selected elements and choose **Add elements to new component**. The newly created Material(2) is a duplicate of the first material but you can change its properties independently.



Step 7

Click on an empty space in the graphics area to clear the selection then click **Show element surfaces**. Notice that each material's component has a different color so you can easily distinguish them.

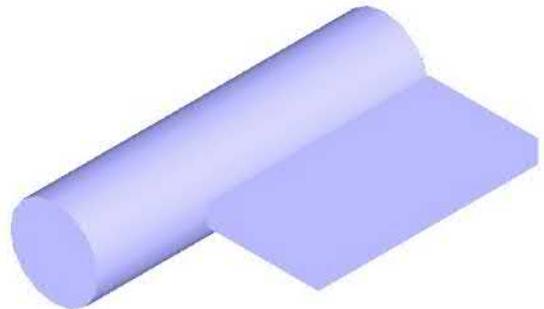


3.9 Mixed Materials with CAD Assemblies

This tutorial shows you how to make a model from multiple parts which may have different materials. To use your own models, you must first export each part from your CAD software as a separate STEP file.

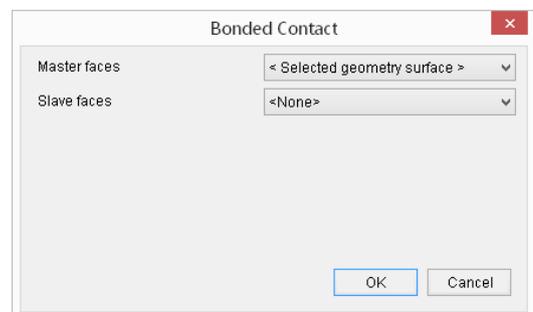
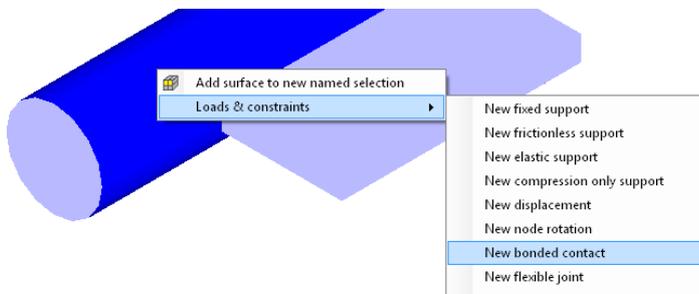
Step 1

Use **File** → **Import** to import **plate.step** and **cylinder.step** from the tutorials folder where Mecway has been installed. For convenience, you can open both files at the same time by holding **Ctrl** while selecting them.



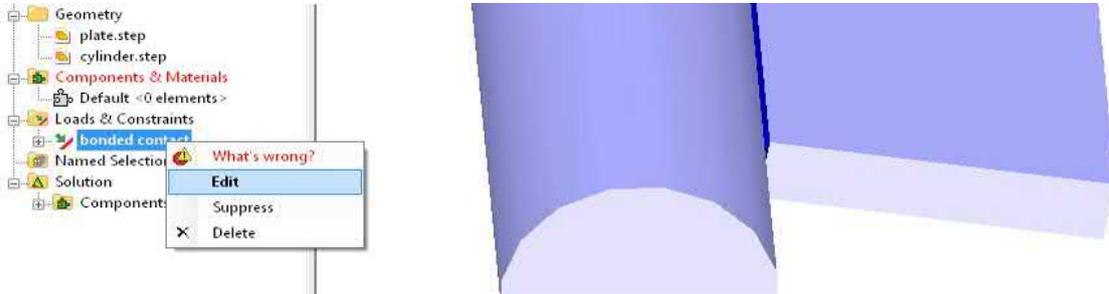
Step 2

Right click the cylindrical surface, then click **Loads & constraints** → **New bonded contact**. Set **Master faces** to **<1 selected geometry surface>** and click OK.



Step 3

Use the middle mouse button to rotate the geometry and find the thin surface of the plate adjacent to the cylinder. Click this surface to select it. Right click **bonded contact** in the outline tree and click **Edit**. Set Slave faces to **<1 selected geometry surface>** and click **OK**.

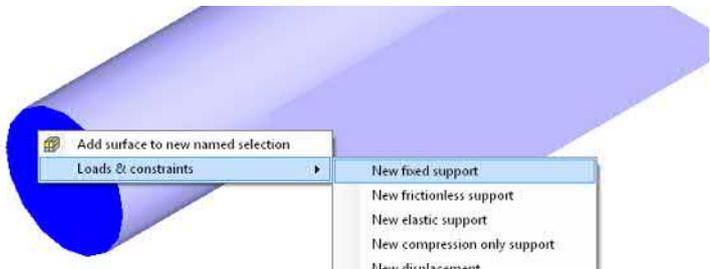


Step 4

Right click **Geometry** in the outline tree and click **Generate all meshes**.

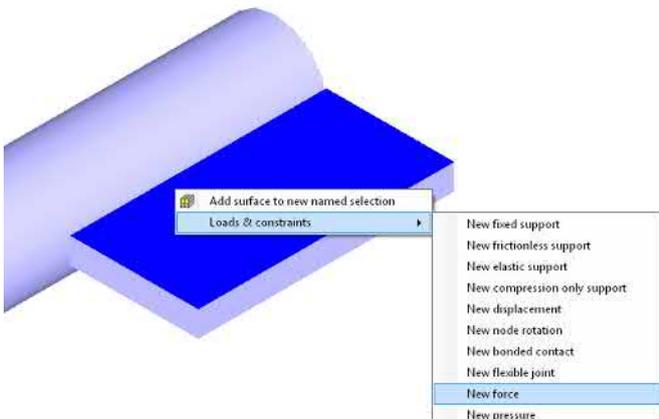
Step 5

Click the **Geometry** branch of the outline tree to return to the geometry view. Right click the circular end of the cylinder and click **Loads & constraints** → **New fixed support**. Click **OK**.



Step 6

Right click the large surface of the plate and click **Loads & constraints** → **New force**. Specify a Y component of -100 lbf. Click **OK**.



Step 7

Right click the **plate <### elements linked>** component in the outline tree, click **Assign new material** and specify the following data:

Isotropic

Young's modulus: 70 GPa

Poisson's ratio: 0.35

Press **OK** and repeat for the **cylinder <### elements linked>** component and the following material data:

Isotropic

Young's modulus: 210 GPa

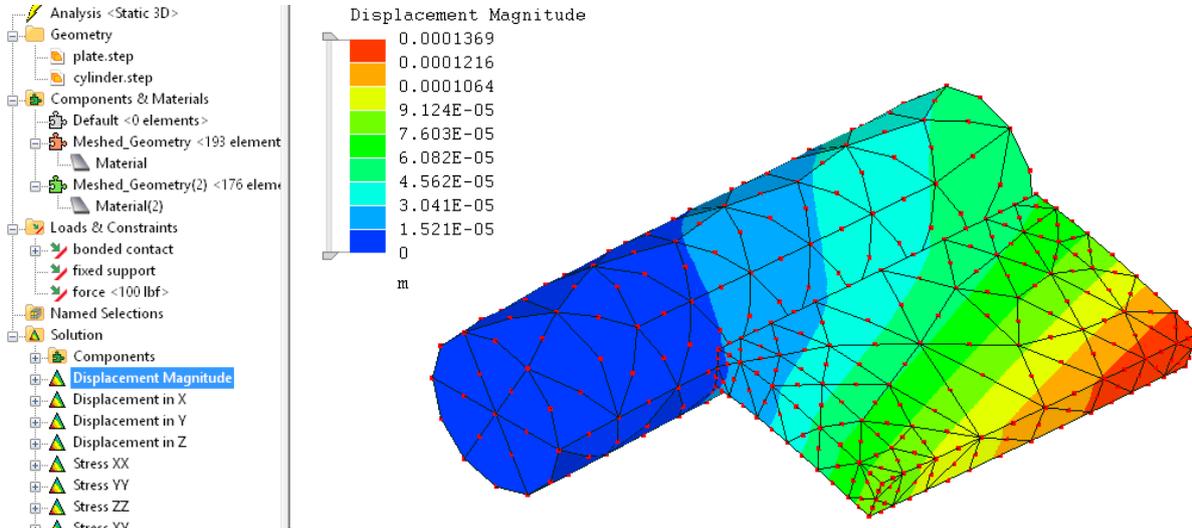
Poisson's ratio: 0.3

Step 8

Click solve .

Step 9

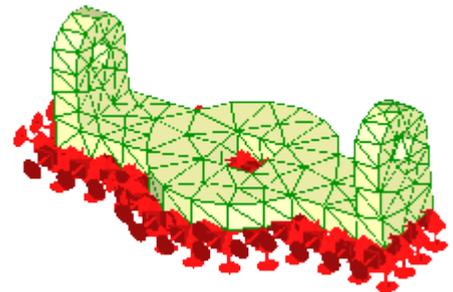
Click the **Displacement Magnitude** field variable then the Animate button  to show the deformation caused by the load.



3.10 Constraint Equations

Step 1

Open **CoupledTutorial.liml** from the tutorials folder where Mecway has been installed.



Step 2

Activate **Select nodes** 

Step 3

Right click **Loads & Constraints**  **Loads & Constraints** and select **New constraint equation**

Click **New term** and specify:

Coefficient 1
DOF Displacement in X
Node 201

Click **OK**

Click **New term** again and specify:

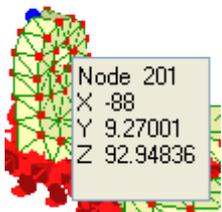
Coefficient -1
DOF Displacement in X
Node 36
 Click **OK**

The constraint equation should now read

$$1 u_{x_{201}} - 1 u_{x_{36}} = 0$$

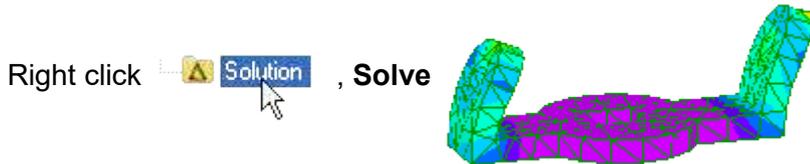
Click OK to save it, then create two more constraint equations in the same way – one with Displacement in Y and one with Displacement in Z.

Step 4



Right click this node and select **Loads & constraints** → **New force**
Apply to <Selected nodes>
X 100

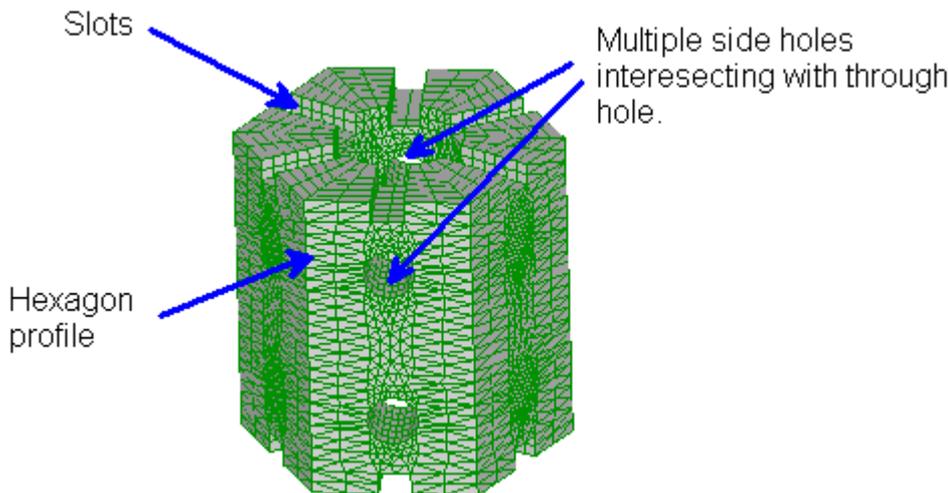
Step 5



The two nodes have the same displacement as if they are connected by a rigid, non-rotating bar.

3.11 Advanced Manual Meshing

Step 1



Identify the most difficult features and model them before the easier features.

In this model the side holes that intersect with the through hole are the most difficult, so this will be modeled first.

Step 2

Mesh tools → **Create** → **Node...**

Next node's coordinates:

X 12
Y 0
Z 0

Click the **Add** button

Change **Y** to 100 click **Add**
change **X** to -12 click **Add**
change **Y** to 0 click **Add**
click **Close**



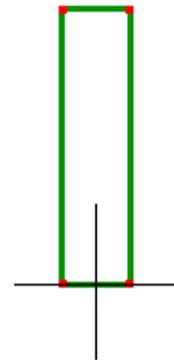
Click the Z arrowhead to view the model parallel to the screen.

View → **Fit to window**

Mesh tools → **Create** → **Element...**

Create a line2 element by clicking on the bottom-right node, then the top-right node. Repeat this to create a total of 4 elements as shown.

For now it has been modeled to a width of 24 but later it will be stretched to the actual width of 50. This was necessary because the left and right faces will be rotated in a later step and we didn't want them to penetrate other elements inside the mesh.



Step 3

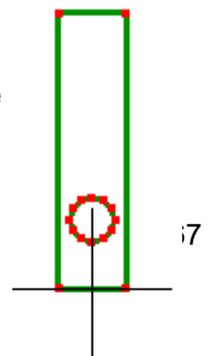
Mesh tools → **Create** → **Curve generator...**



D1 16
D2 16

Click OK to exit the ellipse dialog but don't exit the curve generator dialog just yet.

Change the $Y = 8 \cdot \sin(p)$ into $Y = 25 + 8 \cdot \sin(p)$. This will move the circle up by 25 where we would like it to be positioned. Change the **Number of elements** to 12. Click OK to exit the curve generator dialog.



Step 4

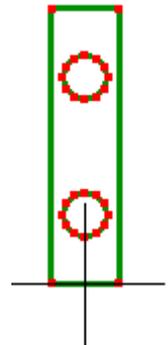
Mesh tools → **Create** → **Curve generator...**



D1 16
D2 16

Click OK to exit the ellipse dialog but don't exit the curve generator dialog.

Change the $Y = 8 \cdot \sin(p)$ into $Y = 75 + 8 \cdot \sin(p)$, this will move the circle up by 75 where we would like it to be positioned. Change the **Number of elements** to 12. Click OK to exit the curve generator dialog.

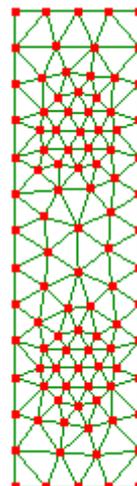


Step 5

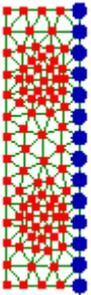
You will be meshing this with the 2D automesh. But before doing so, you have to delete duplicate nodes created during the meshing operations using **Mesh tools** → **Merge nearby nodes** with a **Distance tolerance** of 0.001. Note the change in node numbers in the status bar after this command has been run.

Mesh tools → **Automesh 2D...**
Maximum element size 8

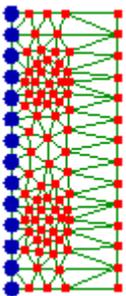
The automesh will briefly run in a separate window and then close.



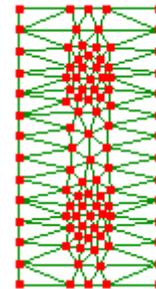
Step 6



Select these nodes. Right click on the selected nodes and choose **Node coordinates** and enter 25 for the **X** coordinate.



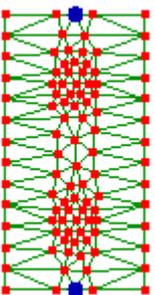
Select these nodes. Right click on the selected nodes and choose **Node coordinates** and enter -25 for the **X** coordinate.



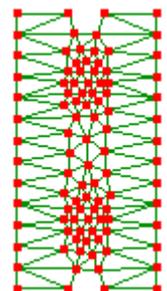
The hexagon edge length has now been restored to 50. It had to be widened because those two faces will be rotated by 30 degrees later.

Step 7

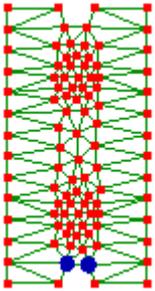
Activate select nodes 



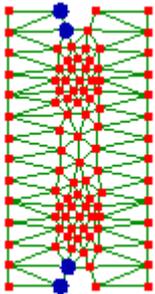
Select the following nodes and press the **delete** key. This is to prepare for the slot that will be done at a later step.



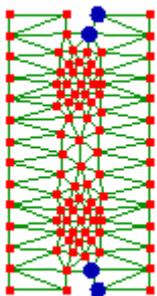
Select these nodes then right click on one of the selected nodes and choose **Node coordinates** and enter 93 for the **Y** coordinate.



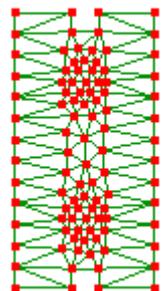
Likewise, select these nodes then right click on one of the selected nodes and choose **Node coordinates** and enter 7 for the **Y** coordinate.



Select these nodes. Hold the **Ctrl** key while selecting them. Right click on one of the selected nodes and choose **Node coordinates** and enter -5 for the **X** coordinate.



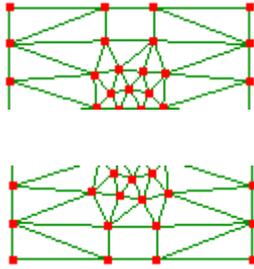
Select these nodes. Hold the **Ctrl** key while selecting them. Right click on one of the selected nodes and choose **Node coordinates** and enter 5 for the **X** coordinate.



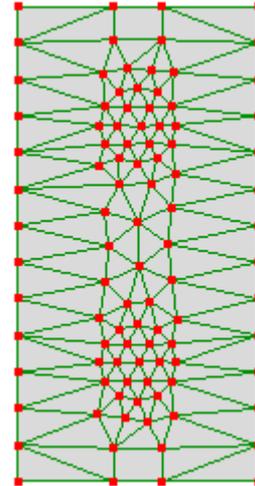
The mesh is now prepared for the 10×7 slot

Step 8

Mesh tools → **Create** → **Element...**
quad4 shell



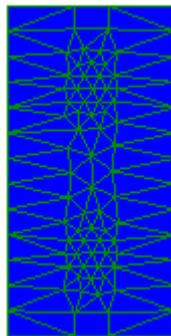
Click the nodes to form quadrilaterals.



Step 9

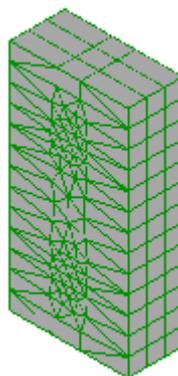
Activate select faces 

Select the entire mesh.



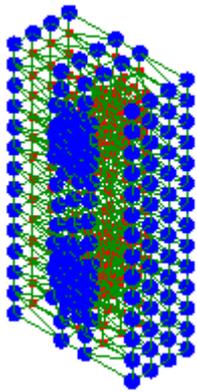
Mesh tools → **Extrude...**

Direction +Z
Thickness 23.3
Number of subdivisions 3

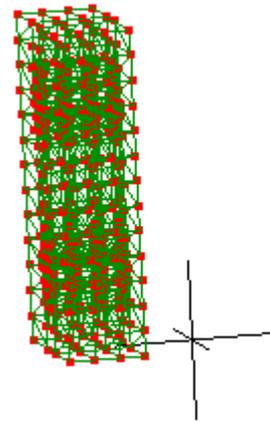


Step 10

Activate select nodes 



Select the entire mesh, **Mesh tools** → **Move/copy...**
Z 20



Step 11

Activate select nodes 

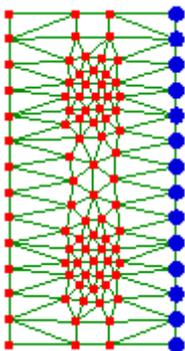


Node 392
 X 25
 Y 0
 Z 43.3

We will be using this co-ordinate information to rotate the right hand face.



Click the Z arrowhead to view the model parallel to the screen.



Select all the nodes on this face.

Mesh tools → **Rotate/copy...**

Axis direction Y

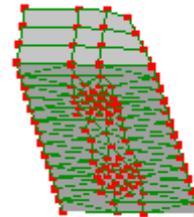
X 25

Y 0

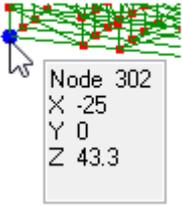
Z 43.3

Angle 30°

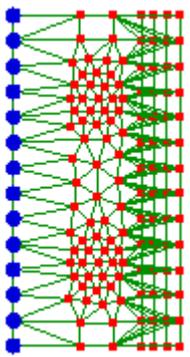
Make sure that the copy check-box is not selected.



Step 12



We will be using this co-ordinate information to rotate the left hand face.



Select all the nodes on this face.

Mesh tools → **Rotate/copy...**

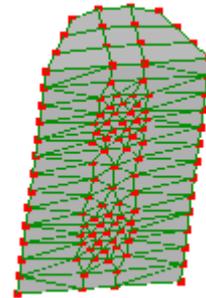
Axis direction **Y**

X -25

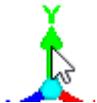
Y 0

Z 43.3

Angle -30°

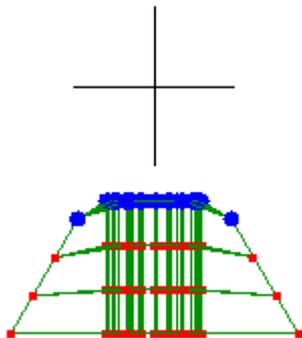


Step 13



Click the Y arrowhead to view the model parallel to the screen.

Activate select nodes 



Select these nodes. **Mesh tools** → **Fit to curved surface...**

Cylinder select

Center

X 0

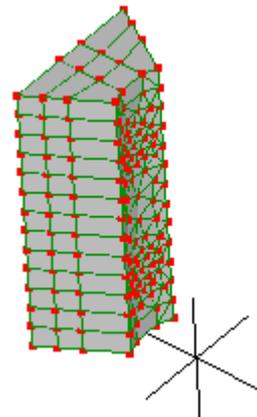
Y 0

Z 0

Radius 20

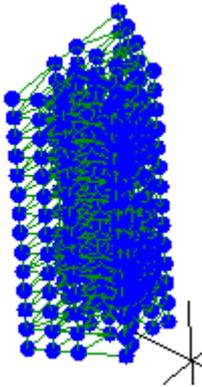
Axis

Y select

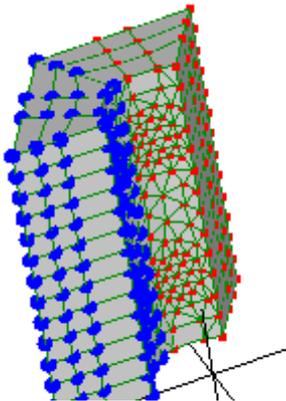


Step 14

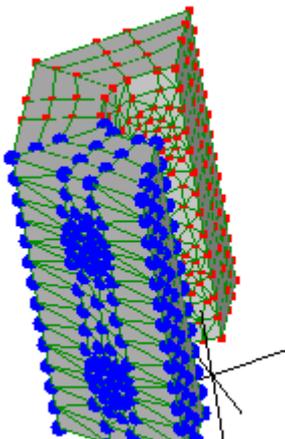
Activate select nodes 



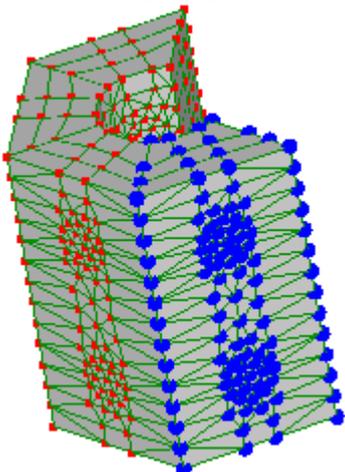
Select the entire model.
Mesh tools → **Rotate/copy...**
Axis direction Y
X 0
Y 0
Z 0
Angle 60°
Copy selected



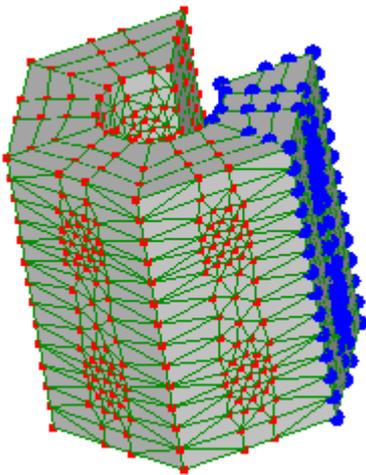
With the newly created elements selected, repeat the **Mesh tools** → **Rotate/copy...** with the same parameters and just click **Apply**.



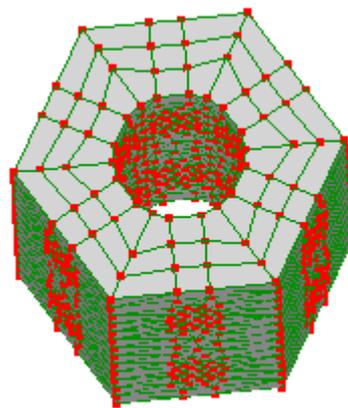
Once again with the newly created elements that are selected, repeat the **Mesh tools** → **Rotate/copy...**



Do it again with the newly created elements that are selected. Repeat the **Mesh tools** → **Rotate/copy...**



Once more for the final time.



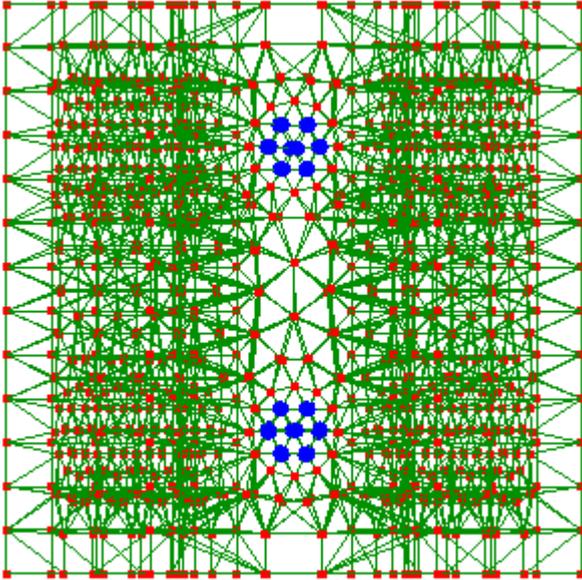
That completes the hexagonal shape. Now use the **Mesh tools** → **Merge nearby nodes** with a **Distance tolerance** of 0.01 to eliminate duplicate nodes created during the meshing operations. Note the change in node numbers in the status bar.

Step 15

Activate select nodes 

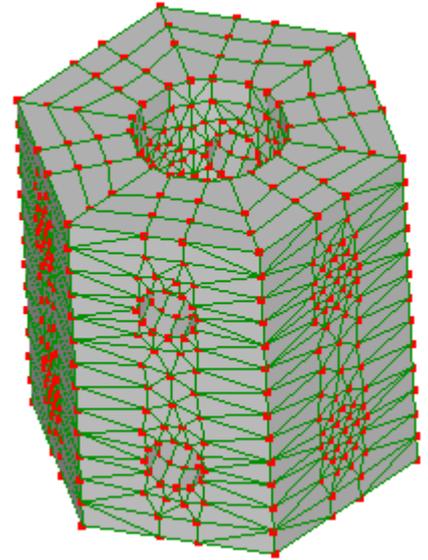


Click the Z arrowhead to view the model parallel to the screen.



Edit → **Circle selection** and drag to select the nodes of the holes. Hold the **Ctrl** key to add to the node selection set. Also hold the **Shift** key to keep a node from moving if you happen to click on it.

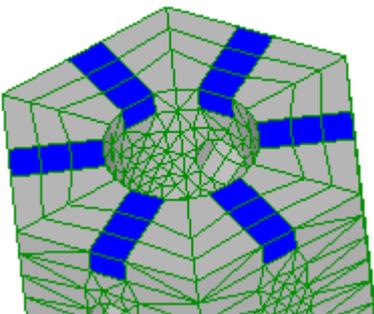
Then press the **delete** key.



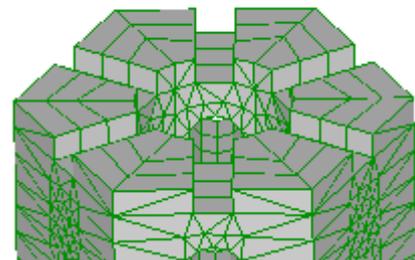
Step 16



Activate select elements

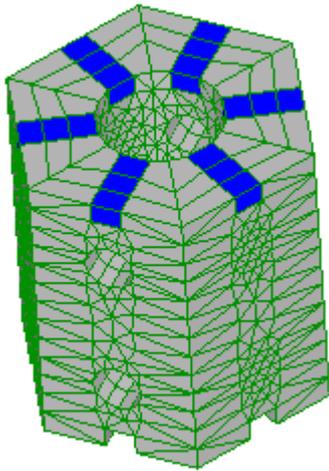


Select the elements where the slots are to be and press the **delete** key.

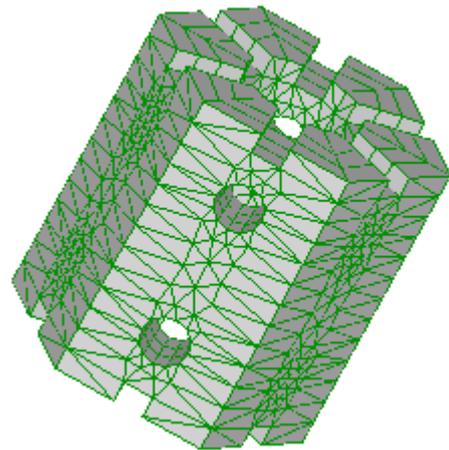


Step 17

 Activate select elements



Similarly, rotate the model and select the elements where the slots are to be and press the **delete** key.



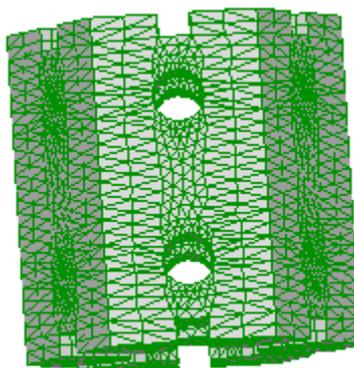
Step 18

As this is a coarse mesh use the **Mesh tools** → **Refine** → **x2** to refine the mesh further.

3.12 Assembly with

This tutorial is to help get a static that are held in place by contact familiar with general use of Mecway

All parts of an assembly must be otherwise the solver won't parts must have constraints to



Contact

solution for an assembly of parts with each other. It assumes you're and contact.

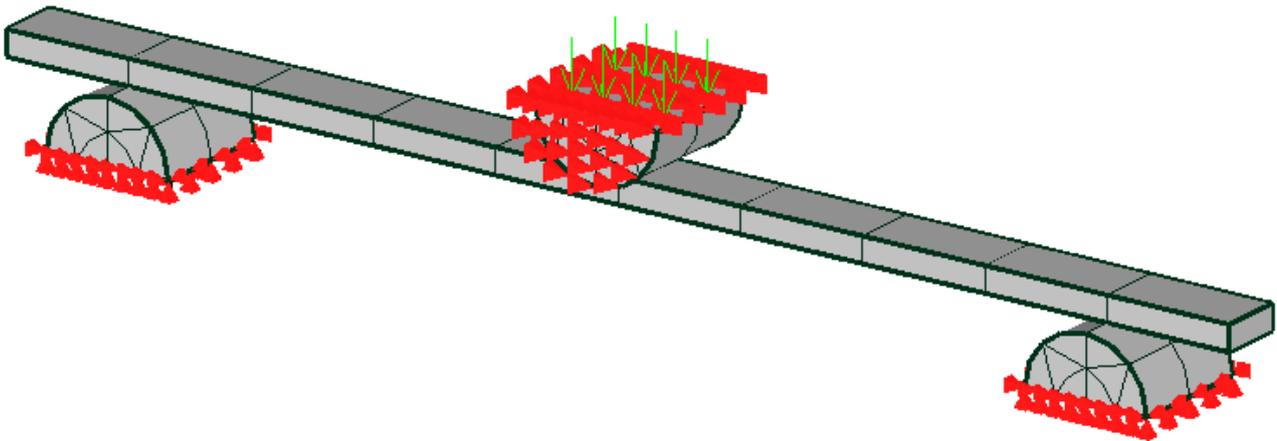
initially in static equilibrium converge. That means all loaded provide reaction forces. Contact is

not reliable as an initial constraint and can leave some parts unconstrained or otherwise prevent the first time step from converging.

We will look at a model of a three-point bending test that fails and try various adjustments until it solves.

Step 1

Open **AssemblyWithContact.liml** from the tutorials folder where Mecway has been installed.



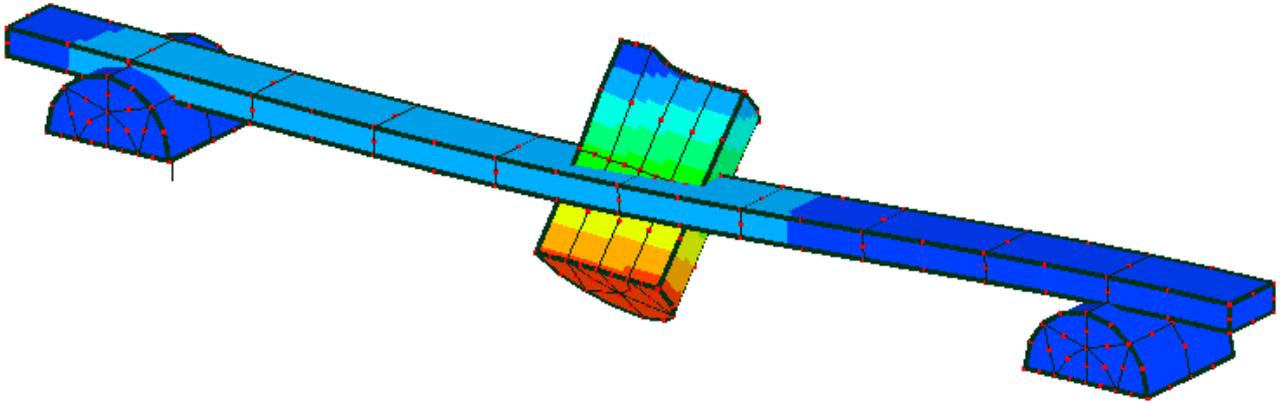
Step 2

Solving the *Naive* configuration. The solver fails with the message

```
*ERROR: too many cutbacks best solution and residuals are in the frd file
```

Step 3

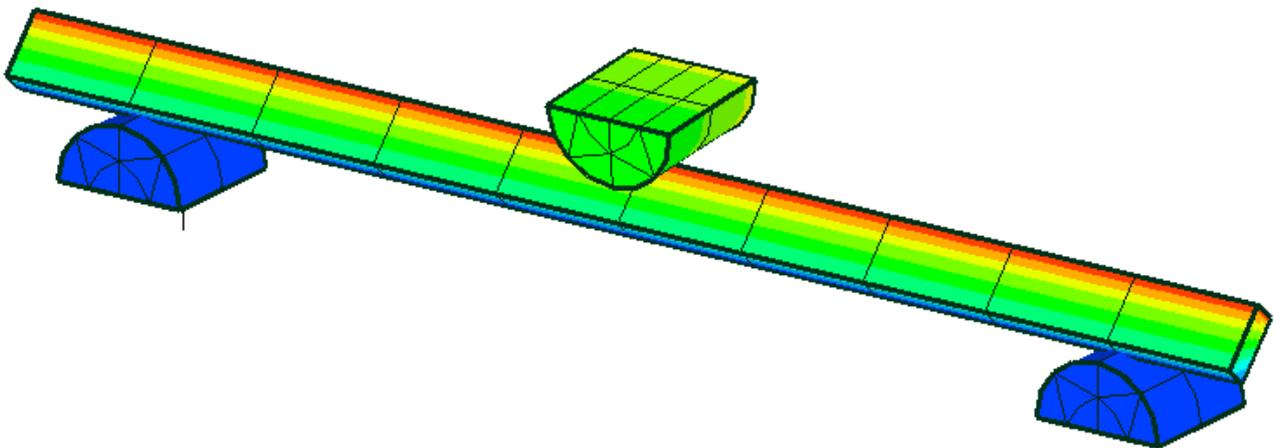
Make sure the problem is caused by rigid body motion and that the contact is otherwise working correctly. We do this by adding a **Fixed support** to each free part. Try this by solving the *Fixed_support* configuration.



It solves quickly and the sample bends into the expected shape. The deformed shape looks strange at first because it's exaggerated due to the high deformation scale factor and low displacement magnitude. This is not the correct solution because of the artificial fixed support but it shows that the failure was likely due to lack of constraints.

Step 4

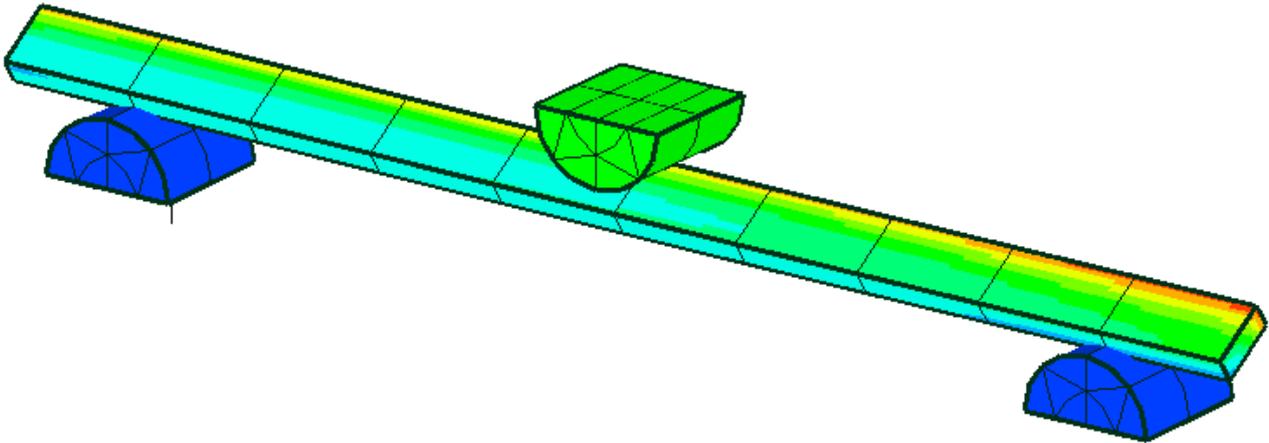
A failure at the first step is physically reasonable because the contacting surfaces are frictionless so the sample is free to slide out of place. We correct this by adding friction to constrain it. Try this by solving the *Friction* configuration.



Unfortunately, friction is not sufficient because the sample and/or loading pin are not being properly constrained by the contacts. You can see this in the non-converged solution. If displacements are too high, set a suitable deformation scale factor in **Solution** → **Deformed view settings**.

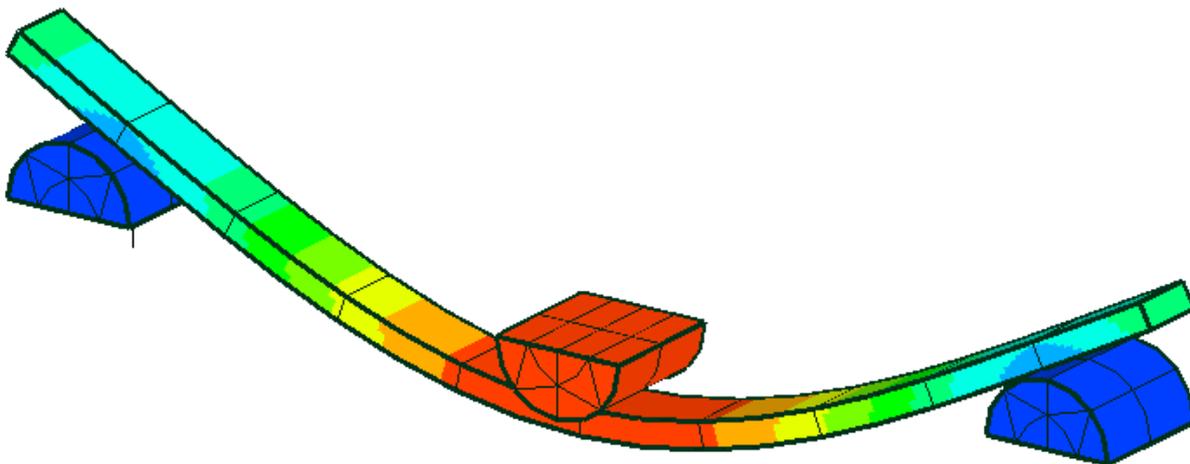
Step 5

We next add **Elastic support** to the sample and loading pin to weakly constrain them. The elastic support's stiffness should be low enough that it won't significantly interfere with the model but high enough to prevent extreme displacements. Try this by solving the *Elastic_support* configuration. It still fails.



Step 6

A final way to help it converge is by starting with a smaller initial force. This is easy to accomplish without affecting the final force by ramping it quadratically ($F_y = -1000 \times t^2$) instead of linearly ($F_y = -1000 \times t$). Try this by solving the *Quadratic_ramp* configuration.



Now it solves successfully. Notice that the displacement is symmetric despite the unsymmetrical locations of the elastic supports. That indicates they are weak enough not to have an obvious effect on

the solution. To measure this more accurately we should check that the external forces at the elastic supports are negligible. Unfortunately, external force is not available with elastic support so you would need to replace it with fixed support connected to the model through elements with a low stiffness material.

That concludes this *Tutorials* guide. Hopefully you will now be quite familiar with Mecway, and have confidence to modify the examples above and build your own models from scratch. For more detailed information and samples, see the companion *Manual*.