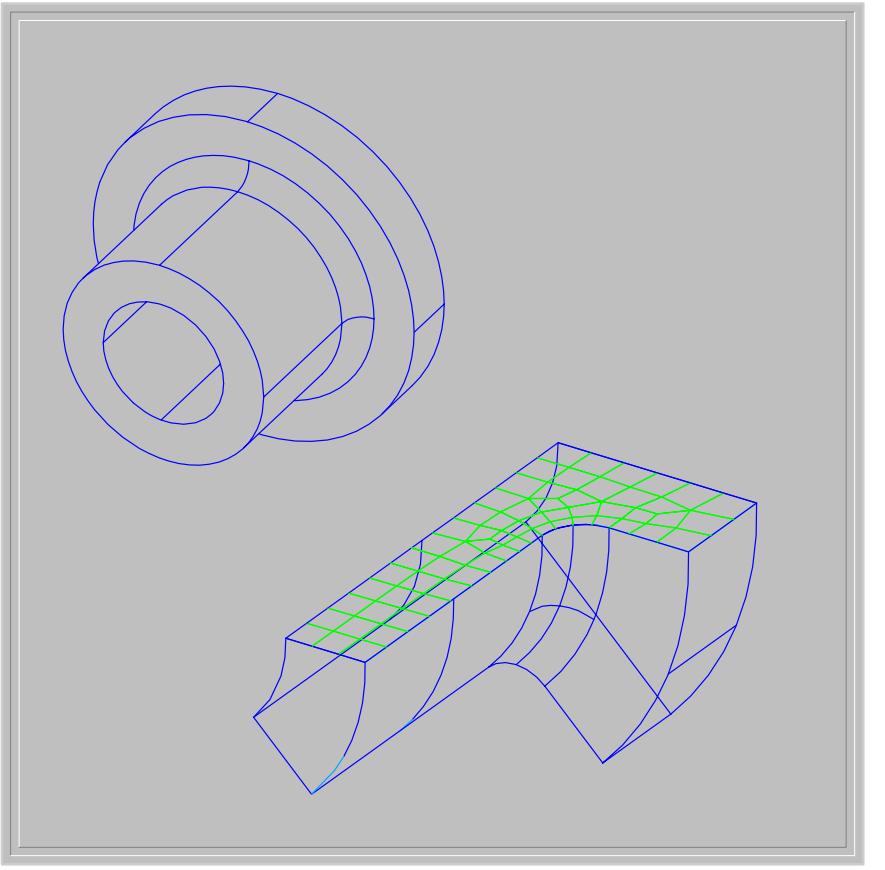


Axisymmetric Modeling

I-DEAS™ Tutorials: Simulation Projects

This tutorial gives an overview of axisymmetric modeling.

Learn how to:

- sketch on the XZ plane
- apply boundary conditions
- mesh axisymmetric elements
- interpret the results

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Quick Tips to Using I-DEAS
  –and–
  Creating Parts
- Introduction to Simulation
- Managing Parts in Model Files
- Free Meshing
- Boundary Condition Sets
- Boundary Condition Surface Loads
- Boundary Condition Symmetry
- Displaying Results
- FE Display Options

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

File
Open

Open Model File form

Model File name: any unique name
OK

Make sure you're in the following application and task:

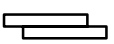
Simulation
Master Modeler

Set your units to mm.

Options
Units

mm (*milli newton*)

Save your model file.



File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:

No

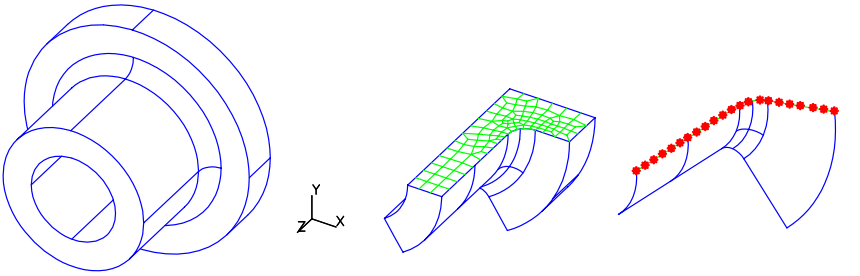
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

Why:
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-Z.

What is axisymmetric modeling?

- Axisymmetric modeling is a way to analyze a revolved part as a 2D model where the part and the loads are axisymmetric. This modeling technique is useful for parts like pressure vessels.
- Axisymmetric elements deform as if each element were a solid ring. Axisymmetric solid elements are modeled like 2D elements. Axisymmetric shell elements are modeled like beam elements.



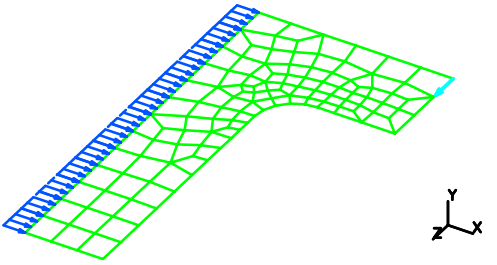
Why use axisymmetric modeling?

- By reducing a 3D model to a 2D plane, there can be orders of magnitude reductions for solution time and file size.

When can you use axisymmetric modeling?

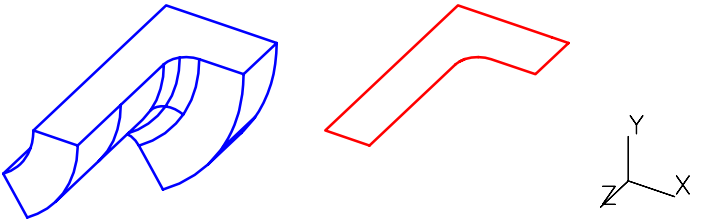
When...

- the part is revolved about an axis
- all loads are also revolved about the same axis, meaning that only ring loads or pressure loads distributed completely around the circumference can be used.



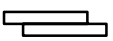
What modeling restrictions are there?

- You must create elements on the part XZ plane such that the Z axis is the axis of symmetry.



What: Turn on the viewport triad display.

How:

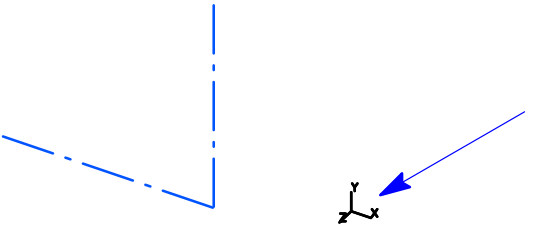

Options
Preferences...


Display...


Viewport Triad

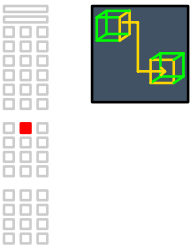

OK (all forms)

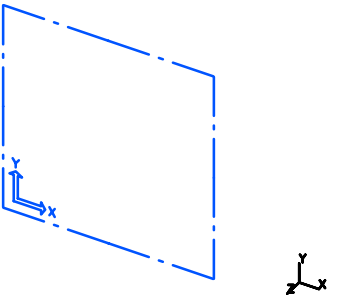
Hint



What: Make sure that the workplane is aligned with the global coordinate system.

Hint



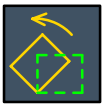



pick workplane


To Global

What: Rotate the workplane to align with the global XZ plane.

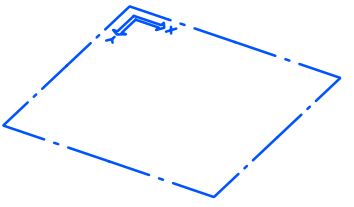
How:



 pick workplane

 *Done*

 *About X*

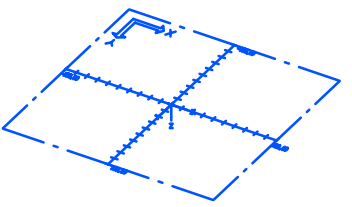


What: Turn on workplane axes.

Hint

 *Display Origin*

 *Display Axes*



What: Sketch a part centerline.

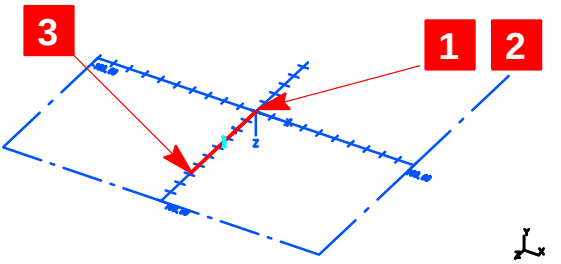
Hint

Focus

1 pick workplane center

2 pick same point as focus point

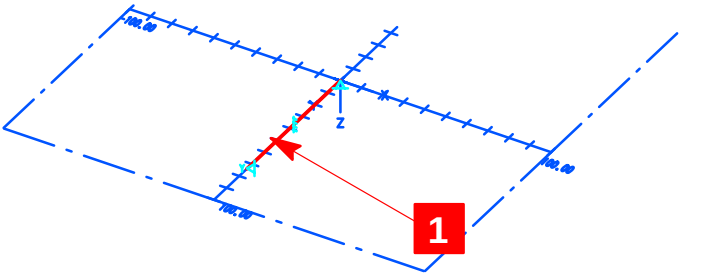
3



What: Anchor this line.

Hint

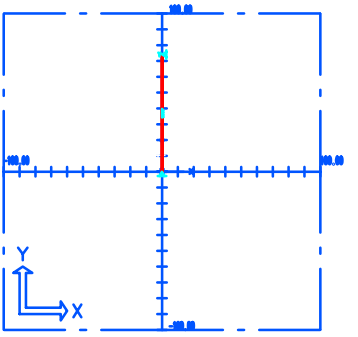
1



Why: The finite element model assumes that the center of rotation is the Z axis. Creating and anchoring this line provides a line to revolve about and a modifiable dimension if you later want to change the radius of the revolve.

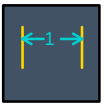
What: Change your view to see the workplane better.

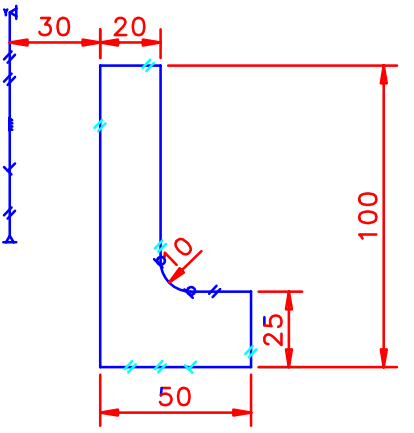
Hint



What: Sketch and dimension the geometry shown.

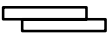
Hint





Why: You must always sketch on the +X side of the plane. The Y axis of the workplane is the axisymmetric centerline (aligned with the global Z axis).

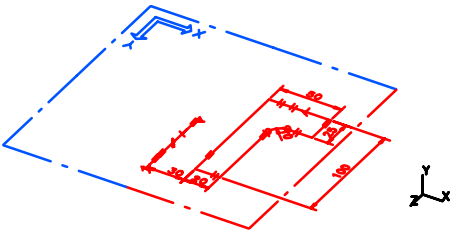
Recovery Point



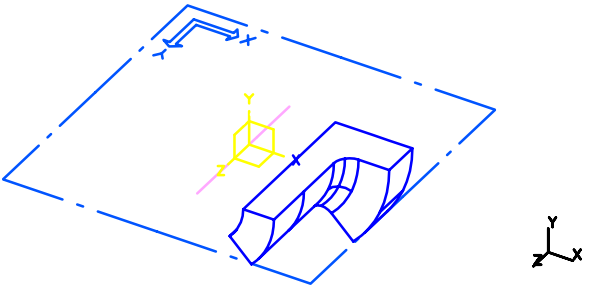
File
Save

What: Revolve the section a negative angle to create a surface for meshing.

Hint



Angle: –45



Why: The angle of rotation must be less than 360 to give a face for meshing. The rotated part segment helps you visualize the true 3D part you're modeling with axisymmetric elements.

Things to notice

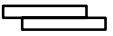
You now have a part with a face for meshing which lies on the part XZ coordinate system.

What: Name the part.

Hint

Name: Revolved Flange

Recovery Point

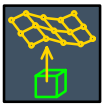


File
Save

What: Create an FE model.

Hint

Boundary Conditions



Geometry Based Analysis Only

What: Restrain one vertex in the Z direction to restrain rigid body motion.

Hint

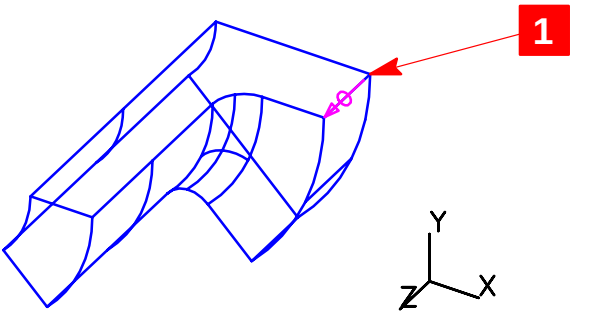
1 pick corner

Specified

Specify Restraint

Z Translation: *constant*

all other translations/rotations: *free*



Why: For axisymmetric elements, there's only one possible rigid body mode, in the Z direction. No restraints are required in the X direction for the solution. (But they may be used.)

What: Apply a force on the edge.

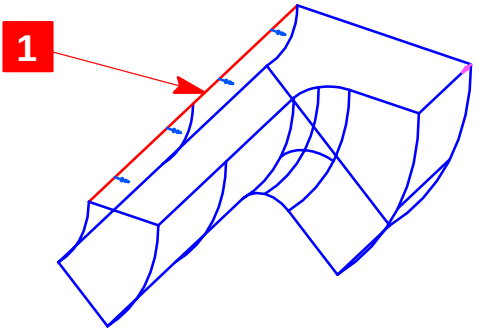
Hint

1 pick edge

pick top face

Axisymmetric Intensity (Force/Area)

In Plane Force: 100



Why: This force on the edge represents a pressure completely around the circumference.

What: Create a boundary condition set containing the loads and restraints.

Hint

Recovery Point

File
Save

What: Mesh axisymmetric solid elements on the face.

Hint

Meshing

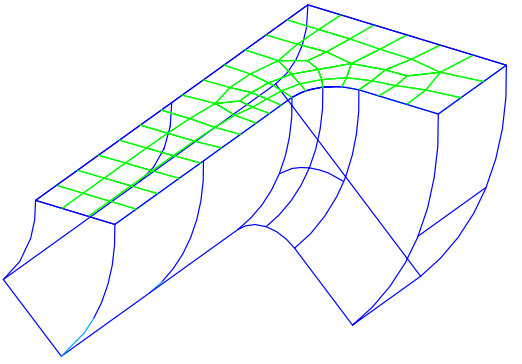
pick surface

Element Family: *Axisymmetric Solid*

Element Length: 8

Free Options...

Percent Deviation: 5



Keep Mesh

Physical properties can use the default null table, since no extra properties are required.

What: Solve the axisymmetric model for linear statics.

Hint

Model Solution

Create...

Linear Statics

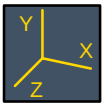
What: Display the stress and deflection.

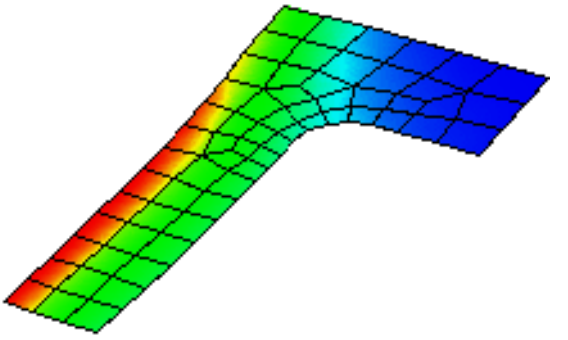
Hint

Post Processing

Create Display form

OK





Things to notice

There is no Y deflection.

Stress components in the X and Z direction are radial and axial. Stresses in the Y direction are “hoop” stresses.

You have completed the Axisymmetric Modeling tutorial.

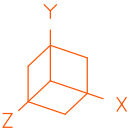
You can delete or put away any FE models or parts. This part is not used in any other tutorials.

Hint

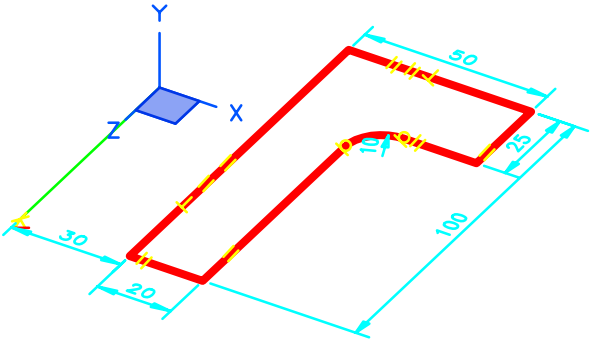
Other methods of modeling on an XZ plane

Another method is to sketch on the XZ plane of the coordinate system of a null part:

- Create a coordinate system and name it to create a null part.



- Sketch on the XZ plane of this coordinate system.



You may also sketch on the XZ plane of the coordinate system of an existing part:

- Display the part coordinate system using *Display Filters*, *Parts*, *Local Origin*.

To mesh an existing part face, the face must be aligned with the part XZ plane.

Aligning elements with the XZ plane

If you've already created elements and they don't lie in the part XZ plane, you may follow these steps to align them.

- Rotate elements using the menu commands

Element
Orient Off Geometry
Rotate


(pick origin)


90,0,0

- List the coordinates of the nodes to make sure the Y value is zero. If not, use:

Element
Orient Off Geometry
Translate

Boundary condition tips

Restraint DOF tips:

- Apply restraints in the XZ plane. (Other directions are not used, and should be left constant.)
- Nodal X and Z restraints represent radial and axial directions.
- Axisymmetric shell elements have an additional rotational DOF about the Y axis.

Loading units:

- The units of nodal force are (force/radian).
  (To apply an axial force of 100, apply (100/(2*Pi))
- Units of element edge in-plane and shear loads are (force/area).
- Gravity vectors must be in the nodal Y direction (along the part Z direction).

Warning!
With versions before Master Series 6, axisymmetric elements did not support geometry-based loads.

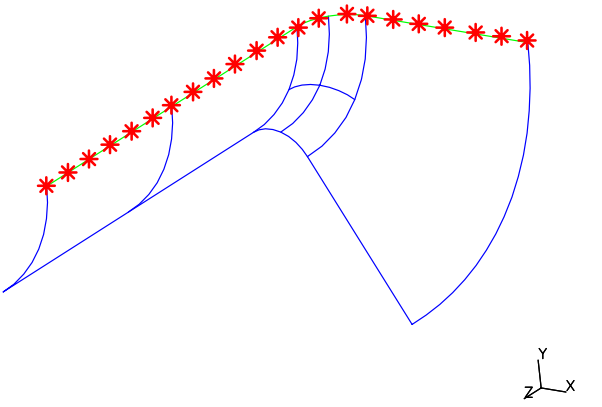
Using axisymmetric shell elements

To mesh axisymmetric shell elements on edges:

Create physical properties for axisymmetric shell elements to define the thickness.

pick edges

Element Family: *Axisymmetric Shell*



Solution tips

Remember to store reaction forces to be able to check the summation of loads.

The mass of axisymmetric elements computed in a verification run is the mass of the entire rotated model, not the mass per radian.

Axisymmetric elements can be used for these types of analyses:

- Linear statics
- Normal mode, Constraint mode dynamics
- Linear buckling
- Nonlinear statics
- Heat transfer
- Potential flow

Warning! Only the axisymmetric modes of vibration or buckling will be obtained, even if other modes would occur first if a complete 3D model were used.

Interpreting results

Deformations in the X and Z directions represent radial and axial deflection. There will be no Y deflection.

Stresses in the X and Z directions represent radial and axial stress. Stress in the Y direction is “hoop” stress.

Tutorial wrap-up

You have completed the Axisymmetric Modeling tutorial.

Delete the part. It is not used in any other tutorial. To delete the part, first delete the finite element model.