Sketching and Constraining

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- sketch using the Dynamic Navigator™
- create constraints
- create dimensions
- modify dimension values and appearance
- add a 2D fillet

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

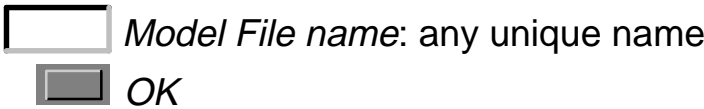   –or–

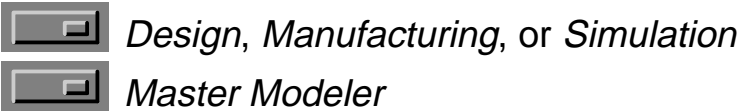   Quick Tips to Using I-DEAS
    –and–
    Creating Parts

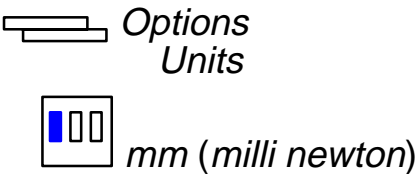If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

⬜️ *File*
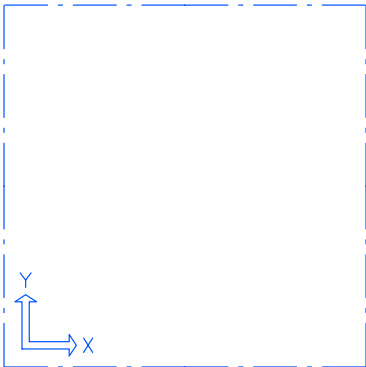   *Open*

Open Model File form

⬜️ *Model File name*: any unique name

⬜️ *OK*

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

⬜️ *Design*, *Manufacturing*, or *Simulation*

⬜️ *Master Modeler*

Set your units to mm.

⬜️ *Options*
       Units

⬜️ *mm* (*milli newton*)

Select *Front View* and *Zoom All*

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.

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.

The Dynamic Navigator recognizes natural engineering relationships (e.g., parallel and perpendicular), and creates constraints to maintain these relationships when geometry or dimensions are changed.

You have used the Dynamic Navigator in previous tutorials. The next few pages show you in more detail how the Dynamic Navigator works.
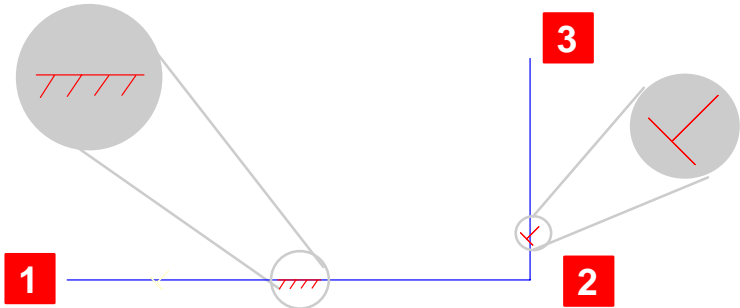
Select the *Polylines* icon.
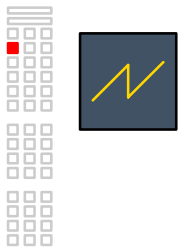
Begin sketching a horizontal and vertical line as shown.
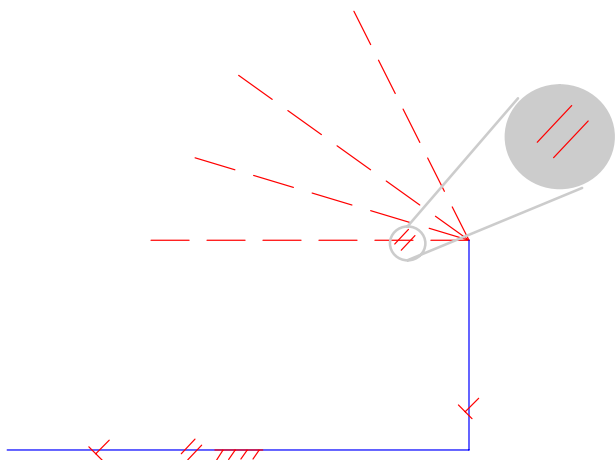
🚫 Don't terminate the command after you sketch.

Things to notice
The first line snaps to horizontal, the second to perpendicular.

Without clicking on a fourth point, move the mouse up and down.

Things to notice

Notice how the software snaps to horizontal, and shows a parallel symbol (parallel to the first line sketched). The Dynamic Navigator's preference is to create constraints to the first geometry sketched.

You can use the *Focus* option to change this order of preference in creating constraints.

[ ] *Focus*

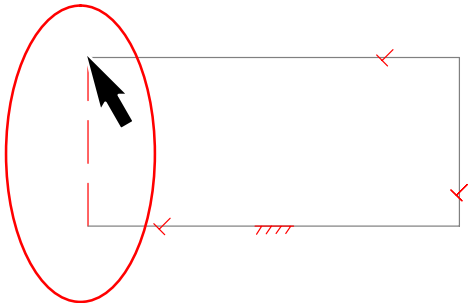**1** pick the vertical line

Move the mouse pointer up and down on the screen.
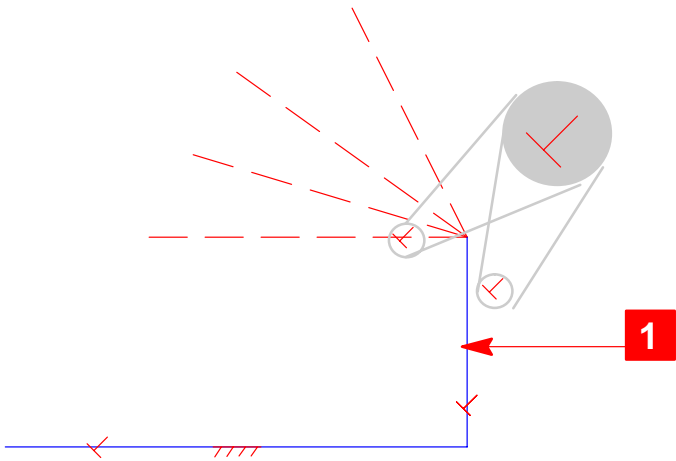
Things to notice
The software now focuses on the vertical line to create a perpendicular constraint instead.

When the cursor is above the first point, there is a dotted vertical projection line aligning the two points.

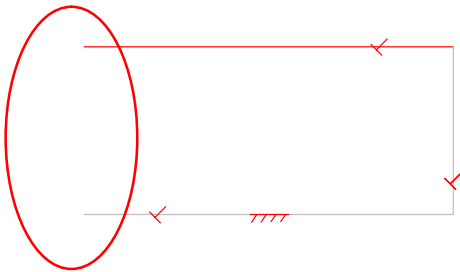Move the cursor above the first point. Notice the dotted line.

Turn off the *Align* option to disable this alignment.

 Align

 Check *I-DEAS List*.
   Notice the message telling you *Align* is off.

Move the cursor over the first point and no dotted line appears.

You can use the options on the right mouse button menu to create an angle of specific degrees.

 Options...

Line Creation Options form

 Angle: 135

 OK

1 click to complete the angled line
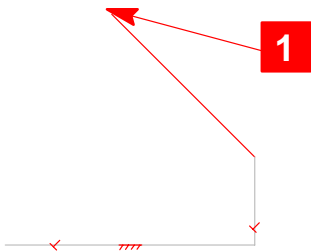
Clear the *Angle* field.

 Options...

Line Creation Options form

 Angle: (clear field using backspace or delete)

 OK

Turn *Align* back on to create the next point aligned vertically with the starting point.

⬚⬚▮ *Align*

1 click on spot above starting point, parallel with first line
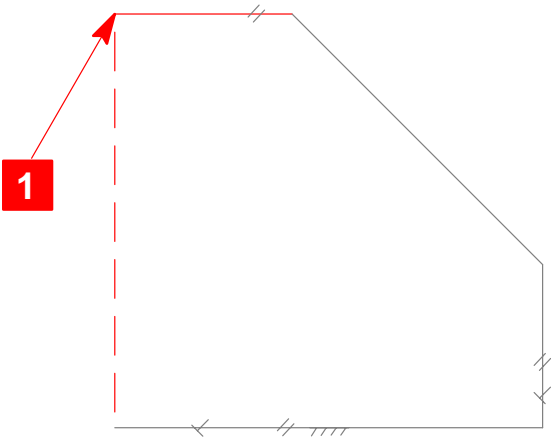
You can also control the Dynamic Navigator constraints with the right mouse button menu.

|▯▯▮| *Navigator*...

Navigator Controls form

🔷 Toggle on every *Recognize* and *Constrain* option, except:

▢ *Angular Dimension* (toggle off if on)

▭ *OK*

Complete the sketch by drawing the last line and clicking on your starting point when you see an x.
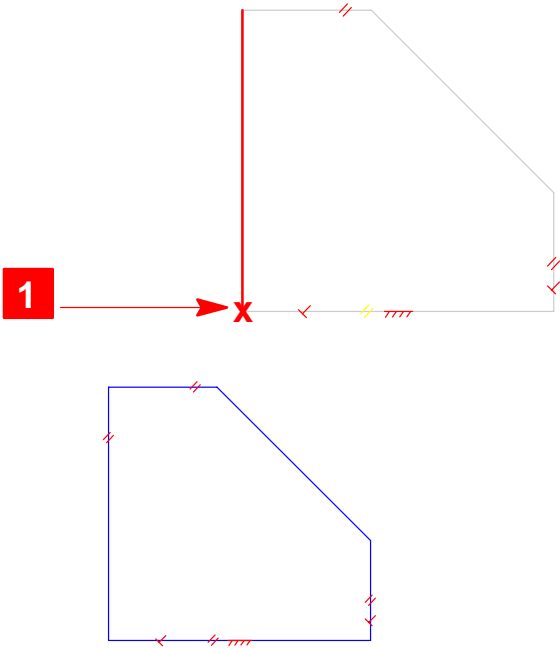
Constraints give you control over how your model incorporates your design intent.

As you just saw, the Dynamic Navigator adds constraints as you sketch. There may be times when you want to create your own constraints.

You can manually create the following constraints:

- anchor
- ground horizontal/vertical
- perpendicular
- parallel
- tangent
- coincident
- collinear

You can also use the *Delete* command to delete manually-created or software-created constraints.

So that you can sketch without using the Dynamic Navigator, delete the entire sketch before continuing.
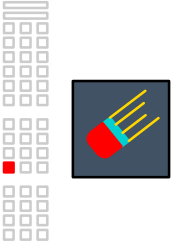
1 click and hold the left mouse button down and drag a box around the sketch

2 release the left mouse button

(Done)

(Yes)

(to deactivate icon)
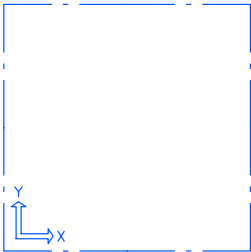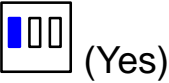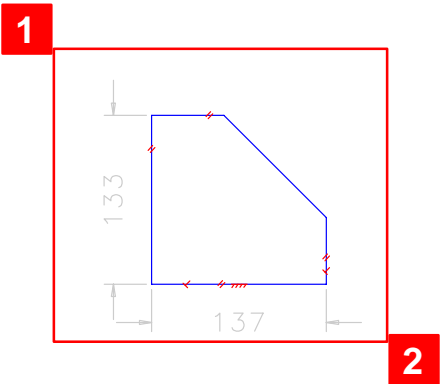
Temporarily turn off all Dynamic Navigator options by holding down the Control key while you sketch the same shape as before.

Warning!
Make sure you release the Control key just before the last point (6), so that the sketch will snap to the starting point.

Hold down the Control key and sketch.
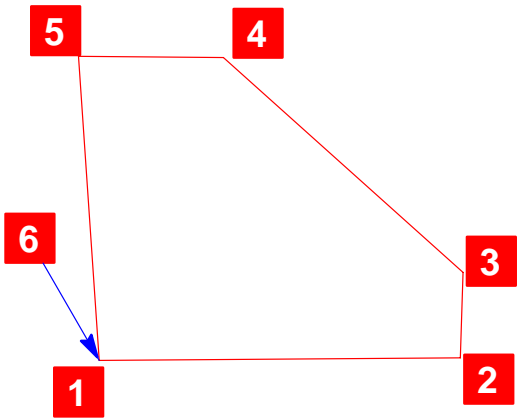
If a constraint, such as parallel or perpendicular, is created on the last line, delete the constraint before continuing.

Hint

 Check *I-DEAS Prompt*.
Make sure you are deleting the constraint and not the geometry.

You can now use icons to create your own constraints.

Anchor the first point.

Create constraints 5 of 7
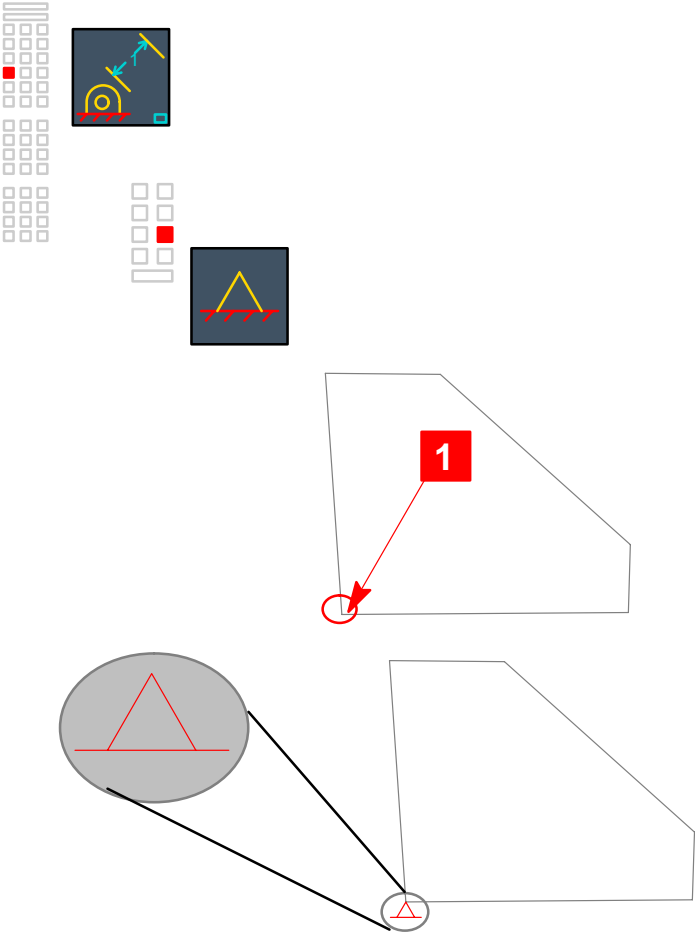
Ground the lower line horizontal.
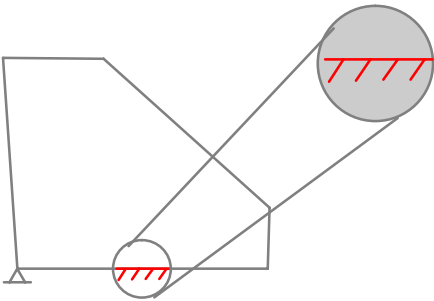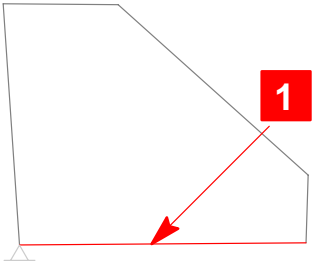
Create a perpendicular constraint between the two vertical lines and the grounded horizontal line.
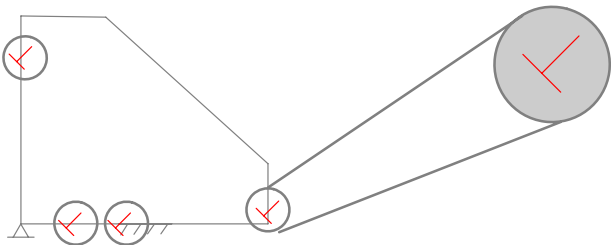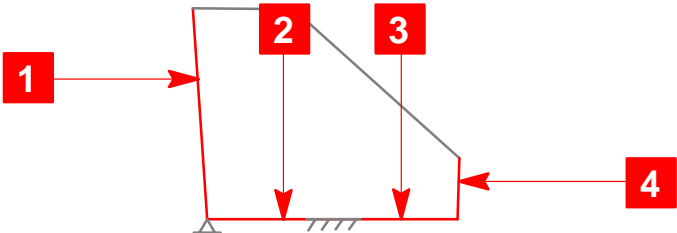
Create a parallel constraint between the top and bottom lines.
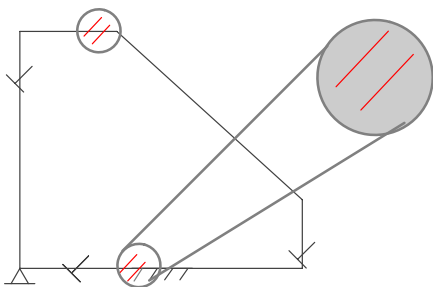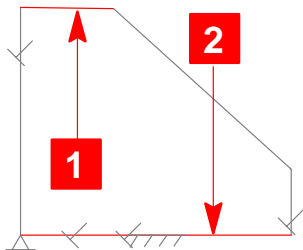
Close the *Constrain* panel.

 Close

Recovery Point
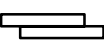
 File
        Save

Dimensions are a special form of constraints. They allow you to control and modify distances between points and lines, while maintaining other geometric constraints and other dimensions.

There are many ways to create dimensions:

- point-to-point
- point-to-line
- line-to-line

With options:

- *Horizontal*
- *Vertical*
- *Flip Arrows*

These options are discussed in this section.

You can modify the display appearance of dimensions after you create them, or you can set the dimension appearance defaults before creating dimensions.

For this tutorial, set the dimension defaults to turn arrows out and to use ANSI standard dimensions.

 Deselect All

 Defaults

 Annotation

Product & Manufacturing Information form

 Arrows: —>||<—

 DIM/GD&T Standard: ANSI 1982

 OK

 You can also set the decimal places' default using this form. For now, leave it set to 0.

Create a line–to–line dimension like the one shown. Don't worry about the actual value. You will modify dimensions later in the tutorial.
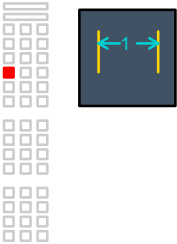
1 pick line

2 pick line

3 place the text
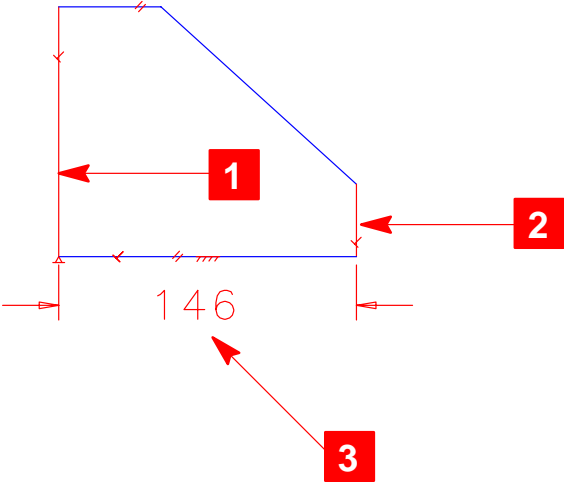
Create a point–to–point dimension, using the *Horizontal* option.
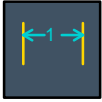
1 pick point

2 pick point

 Horizontal

3 place the text
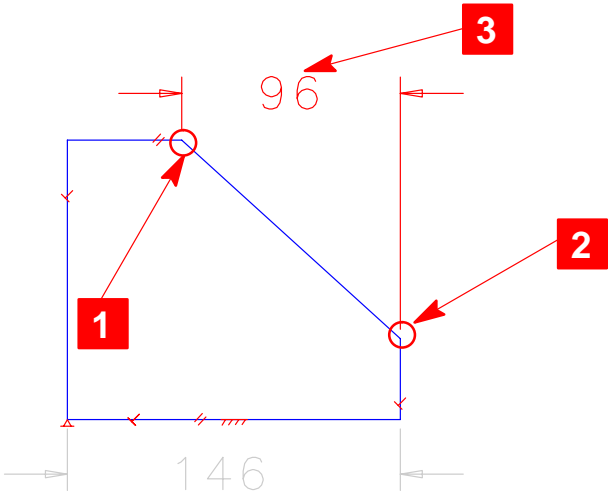
Things to notice
As you add dimensions, the colors will change:
- green = unconstrained
- yellow = partially constrained
- blue = fully constrained

The *Horizontal* and *Vertical* options allow you to control point-to-point dimensions by selecting the horizontal or vertical component of the angled distance.

Create the two vertical dimensions as shown. Create these by picking a line and a point both times.
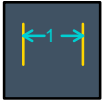
1 pick point

2 pick line

3 place the text
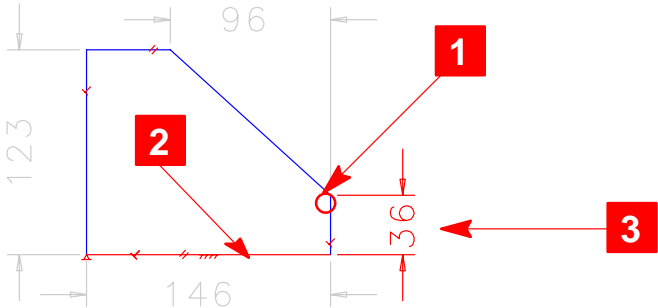
4 pick point

5 pick line

6 place the text
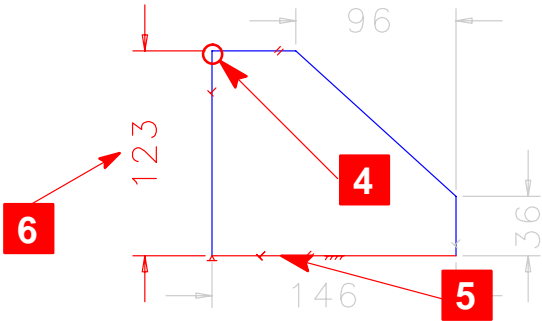
Delete the dimension on the angled face so you can add an angle dimension instead.
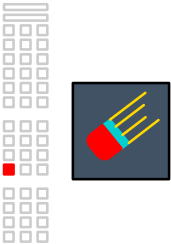
1 pick dimension

 (Done)

 (Yes)

 (to deactivate icon)
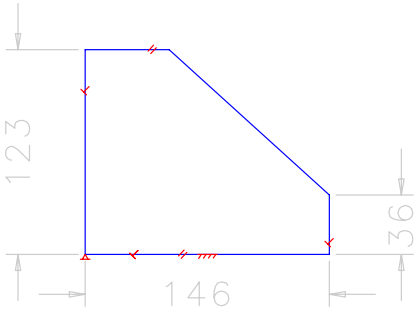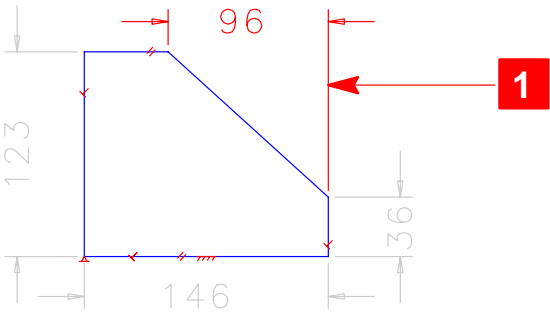
Now, add the angle dimension.
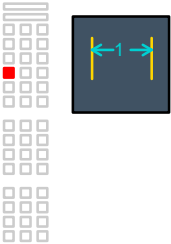
1 pick line

2 pick line

Flip Arrows

3 place the text
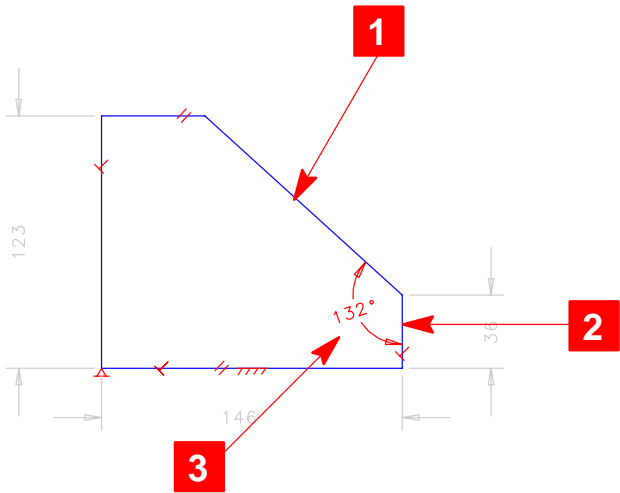
Recovery Point
File
     Save

One of the benefits of I-DEAS is that you can sketch the shape and then later modify the dimensions to the size required.

In this section, you learn different ways to modify dimension values and change their appearance.

You can modify dimensions with the:

- *Drag* command
- *Modify* command

You can change the following appearance of dimensions:

- standards: ANSI or ISO
- decimal places
- arrows (in/out)

Use the *Drag* icon to change the angle to approximately 135 degrees.
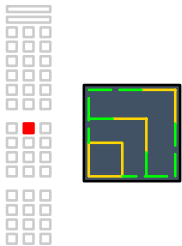
1 pick arrowhead
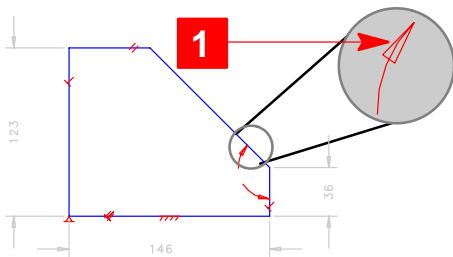
Hint
After picking the arrowhead, move the angled line while watching the degrees in the odometer in the graphics window. When it reaches anywhere between 134/136, click the left mouse button.
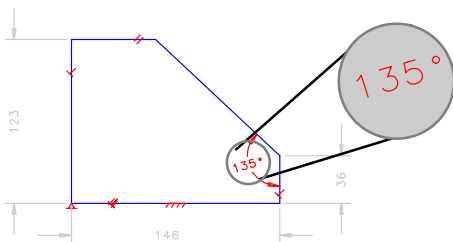
 (to deactivate icon)

Use the *Modify* icon to directly modify the two vertical dimensions as shown.
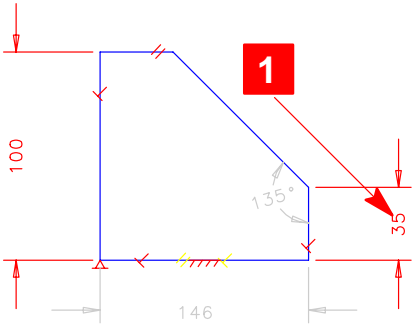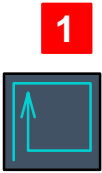
1

Modify Dimension form

⬜ = 35

🔲 *OK*

2

Modify Dimension form

⬜ = 100

🔲 *OK*
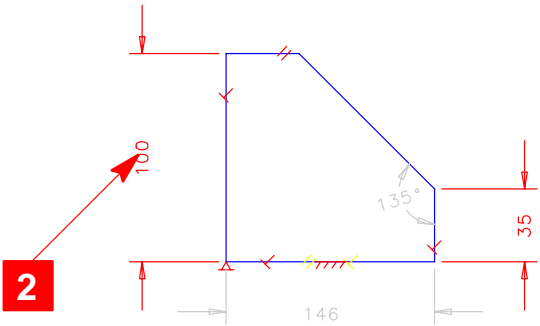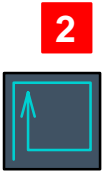
Use the *Modify* icon to match the horizontal dimension to the longer vertical dimension.

[□□▮] *Deselect All*

1

Modify Dimension form

→ *Match*

2

▭ *OK*

[▯▮▯] (Done)

2 →

1
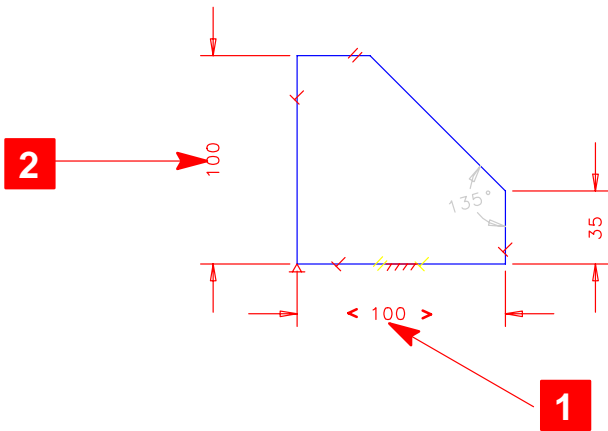
Things to notice
The horizontal dimension that is now matched to the vertical dimension is identified with brackets drawn around it.

Select the two 100mm dimensions and flip the arrows.

1

2 shift-pick
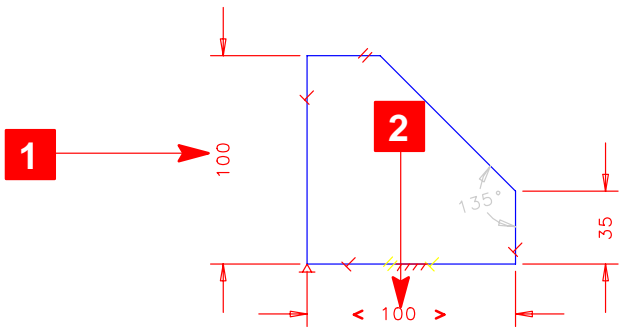
Product & Manufacturing Information form

☐ *Arrows*

☐ *|<– –>|*

☐ *OK*
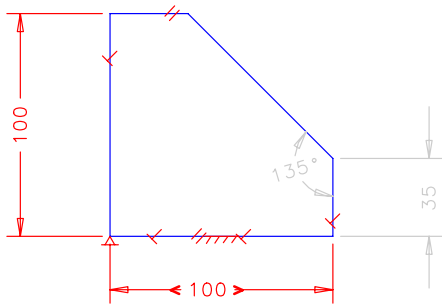
Change the number of decimal places on the 35mm dimension to 2 places.

1 pick the dimension
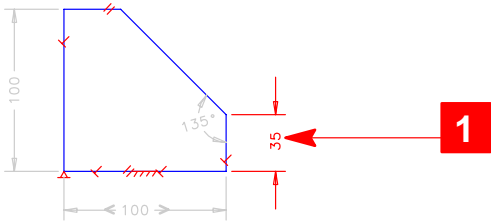
Product & Manufacturing Information form

Units/Decimal Places...

Units & Decimal Places form

Decimal Places: 2

OK

OK
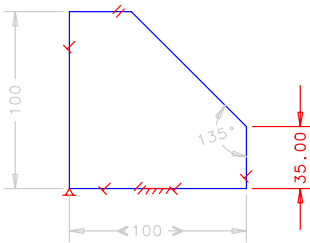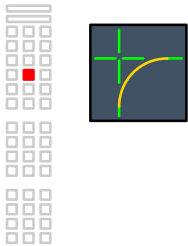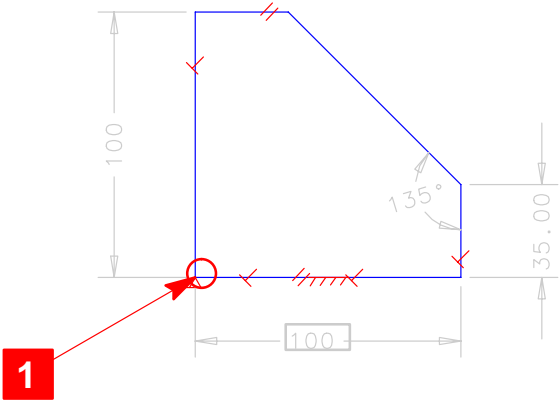

Things to notice
The dimension value is now displayed to 2 decimal places.

You can set the default for any additional dimensions that are created. For example, set the default to two decimal places.

 Deselect All

 Defaults

 Annotation

Product & Manufacturing Information form

 Units/Decimal Places...

Units & Decimal Places form

 Decimal Places: 2

 OK

 OK

Recovery Point
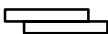
 File
   Save

When you create a fillet, it is automatically constrained with a tangent constraint on both ends and a radius dimension. However, you can control the radius and optionally leave the untrimmed curves in place. These untrimmed curves are useful for dimensioning to the "theoretical corner."

Add a fillet in the lower corner. Keep the untrimmed curves.

1 pick corner

Fillet form

    Radius: 20

    Trim/Extend (toggle off)

    OK
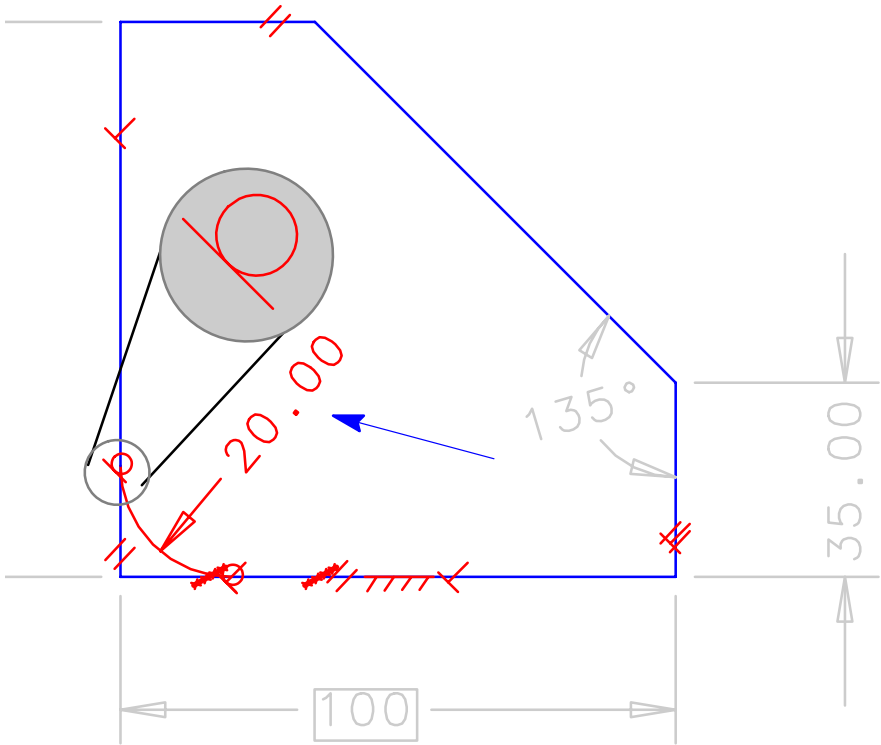
Things to notice

Notice that the radius has two decimal places (since you set the default in the previous exercise).

(to deactivate icon)

Things to notice

Constraints (tangent) and dimensions (radius) are automatically created.

Tutorial wrap-up

You have completed the Sketching and Constraining tutorial.

Make sure you save your model file. The sketch you just created is used in a later tutorial.

    File
        Save

See also...

For additional information on many of the concepts covered in this tutorial, see the following in the I-DEAS *Help* facility:

    Help, Manuals, Table of Contents

Design User's Guide
      Design Concepts
        Modeling Parts

Design Reference Guide
    Master Modeler
      Wireframe Geometry

What's next?

After exiting, choose the Fundamental Skills tutorial that is next in the learning path you are following.

To practice the skills taught in this and preceding tutorials, try the following Advanced Projects tutorial:

- Creating Basic Machine Parts

To exit this tutorial, select:

    File
        Exit

Warning!

Do not use the menu in the *I-DEAS Icons* window to exit. Use the menu in the Acrobat Reader window.

I-DEAS Master Series™ Online Tutorials