Getting Started

Before getting into the detailed instructions for using **Interactive Drafting workbench**, the following tutorial aims at giving you a feel of what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities. You just need to follow the instructions as you progress along.

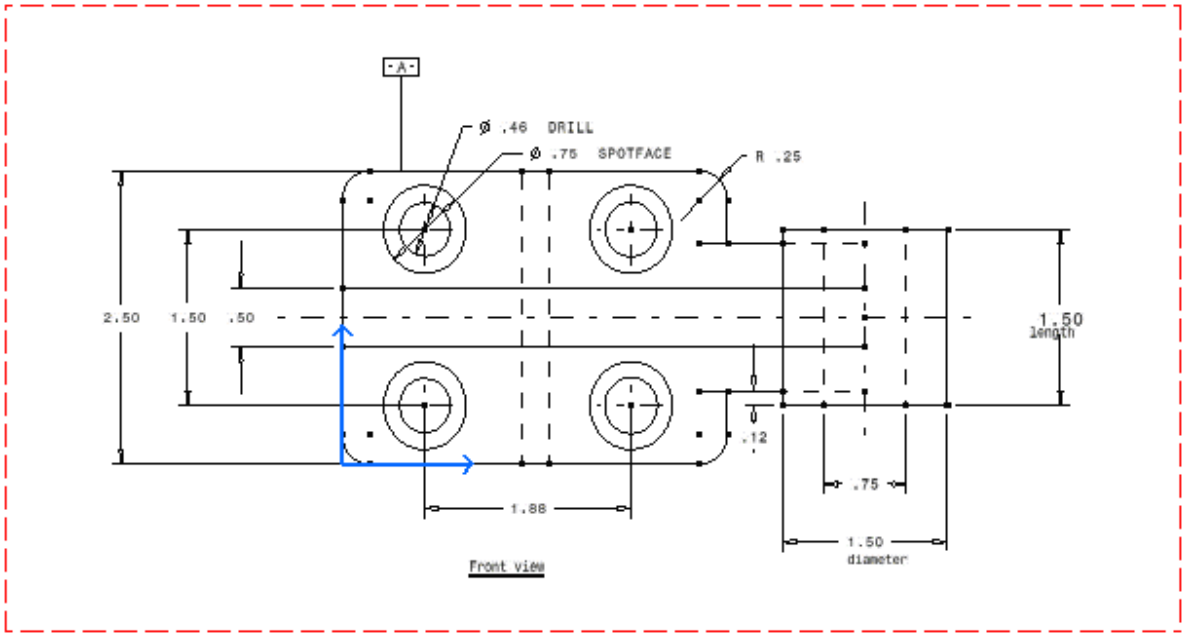
The main tasks described in this section are the following:

Entering the Interactive Drafting Workbench
Creating a New View
Creating a Rectangle
Creating Corners
Creating Lines
Translating Lines
Creating Circles
Creating Dimensions
Creating Annotations

Before discovering this scenario, you should be familiar with the basic commands common to all workbenches. These are described in the *Infrastructure User's Guide*.

All together, the tasks should take about 30 minutes to complete.

The final drawing will look like this:



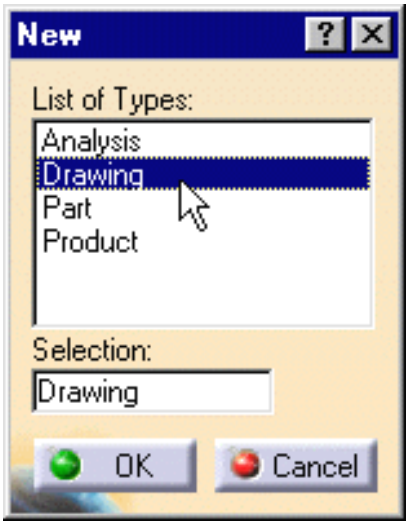
Set the options in `Tools -> Options -> Mechanical Design -> Drafting`: setting the options properly improves the software performances (see Customizing).

Entering the Interactive Drafting Workbench

This first task shows you how to enter the **Drafting workbench** and start a new drawing.

1. Select the `File -> New` command (or click the New icon).

The New dialog box is displayed, allowing you choosing the type of the document you need.



2. Select `Drawing` in the `List of Types` field and click `OK`.

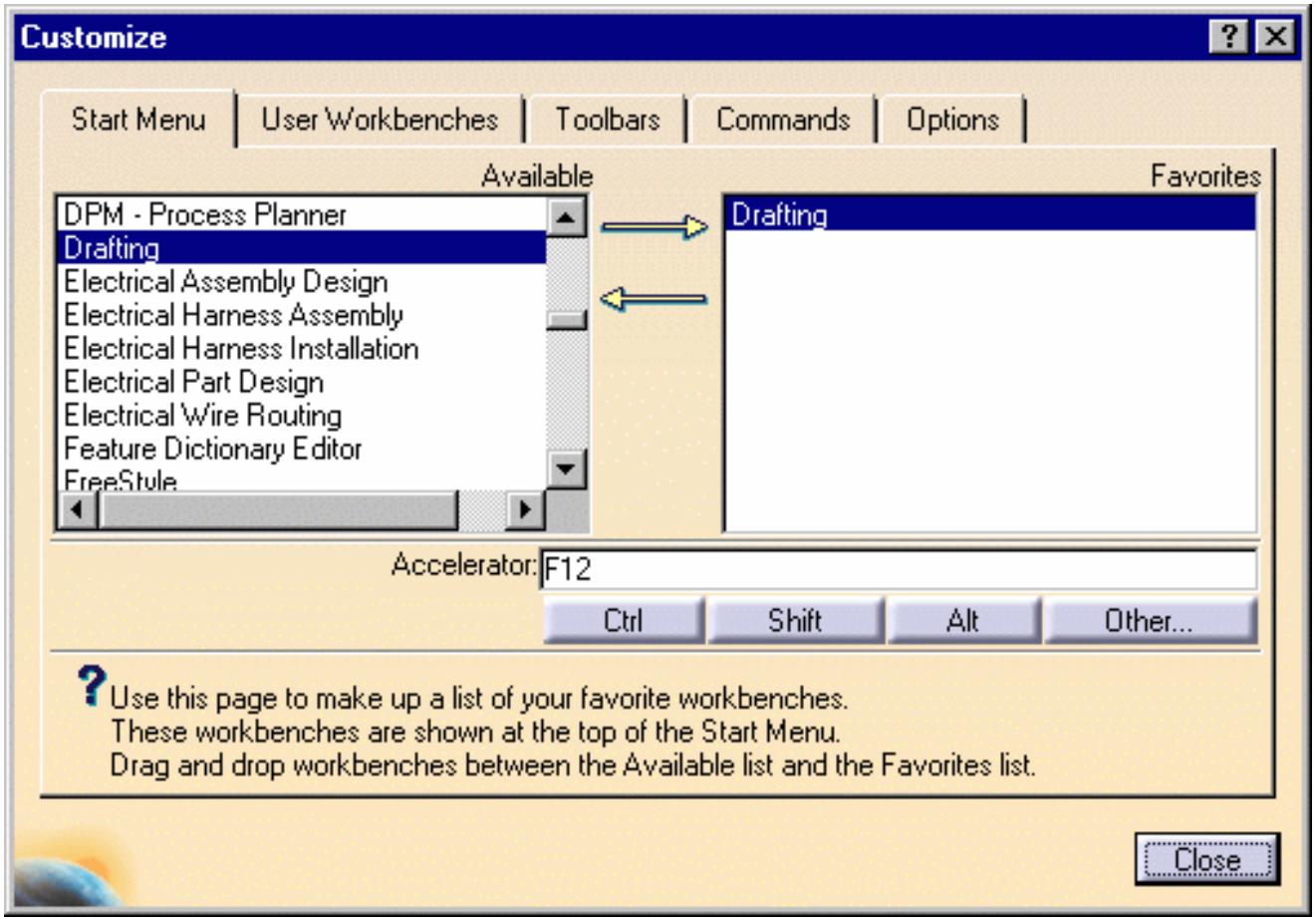
OR

1. Select the `Start -> Mechanical Design` commands from the menu bar.

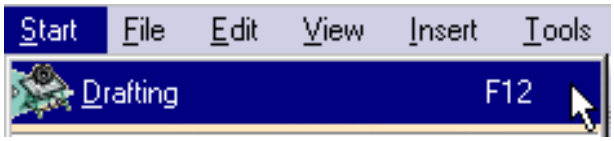
2. Select the `Drafting` workbench.

OR

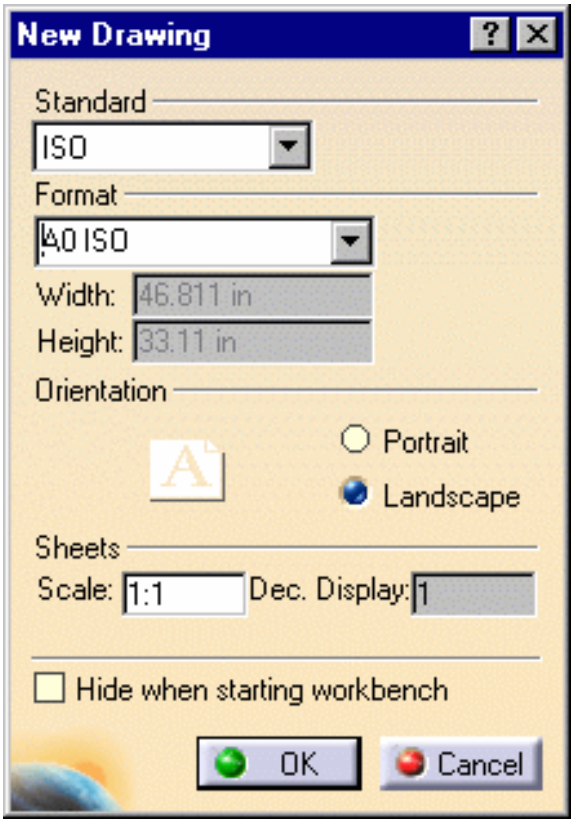
1. Select the `Tools -> Customize` commands (`Start Menu` tab) and define the `Favorites (Drafting)` and `Accelerator (F12)` options as shown below and click the `Close` switch button.



2. Press `F12` key or select the `Start -> Drafting F12` commands from the menu bar.



Whatever the dialog you used for entering the **Drafting workbench** you used, the New Drawing dialog box is displayed, allowing you choosing the type of Standard, Format, Orientation and scale you need.



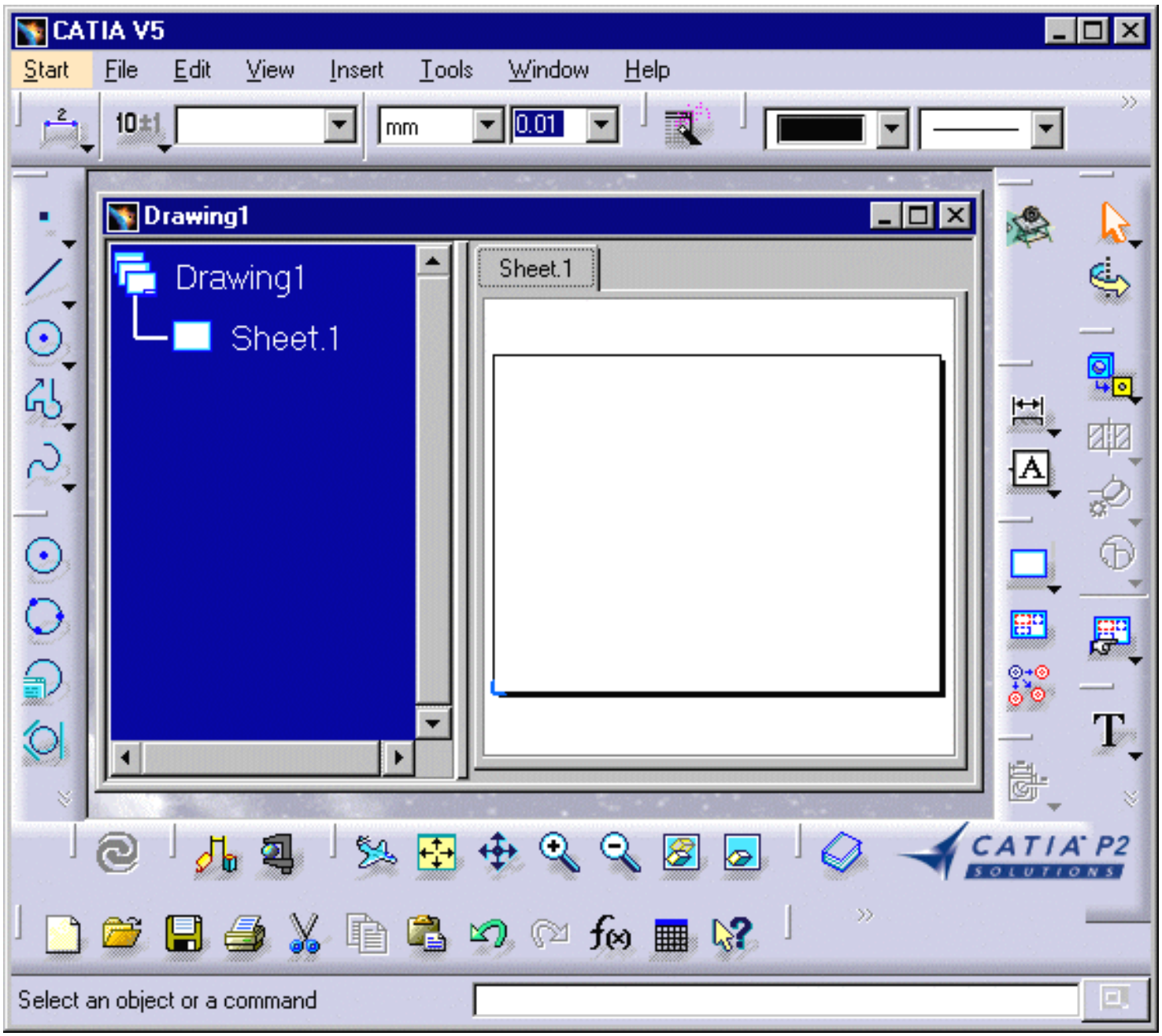
3. Select the `ISO` standard and click the `Landscape` option.

If you activate the `Hide when starting workbench` option, the next time you enter the **Drafting workbench** via `Start -> Drafting F12` or by pressing `F12` key, the New Drawing dialog box will not appear any more. Still, you will always be able to access this dialog box by selecting `File -> New Drawing` command from the menu bar.

4. Click `OK`.

- You can add an unlimited number of customized standards using Standard files that you will create and/or, if needed, modify. Once created, this standard will appear in the New Drawing dialog box. For more details on standards, see the Standards Administration section. Care that any user-defined standard is based on one of the four international standards (ANSI, ISO,ASME or JIS) as far as basic parameters are concerned.
- You can create your own Format:
  - key in the format name in the `Format` field,
  - use the tab key to access to the `Width` and `Height` fields and sets their values.

The **Drafting** workbench is loaded and an empty Drawing sheet opens:



Make sure you customized the units accordingly. For this:

1. Select the `Tools -> Options` command to display the `Options` dialog box.
2. Click General in the list of objects to the left of the Options dialog box.
3. Select the Units tab and set Length to Inch and then click OK.

To visualize better your drawing, tile the windows horizontally from the menu bar.

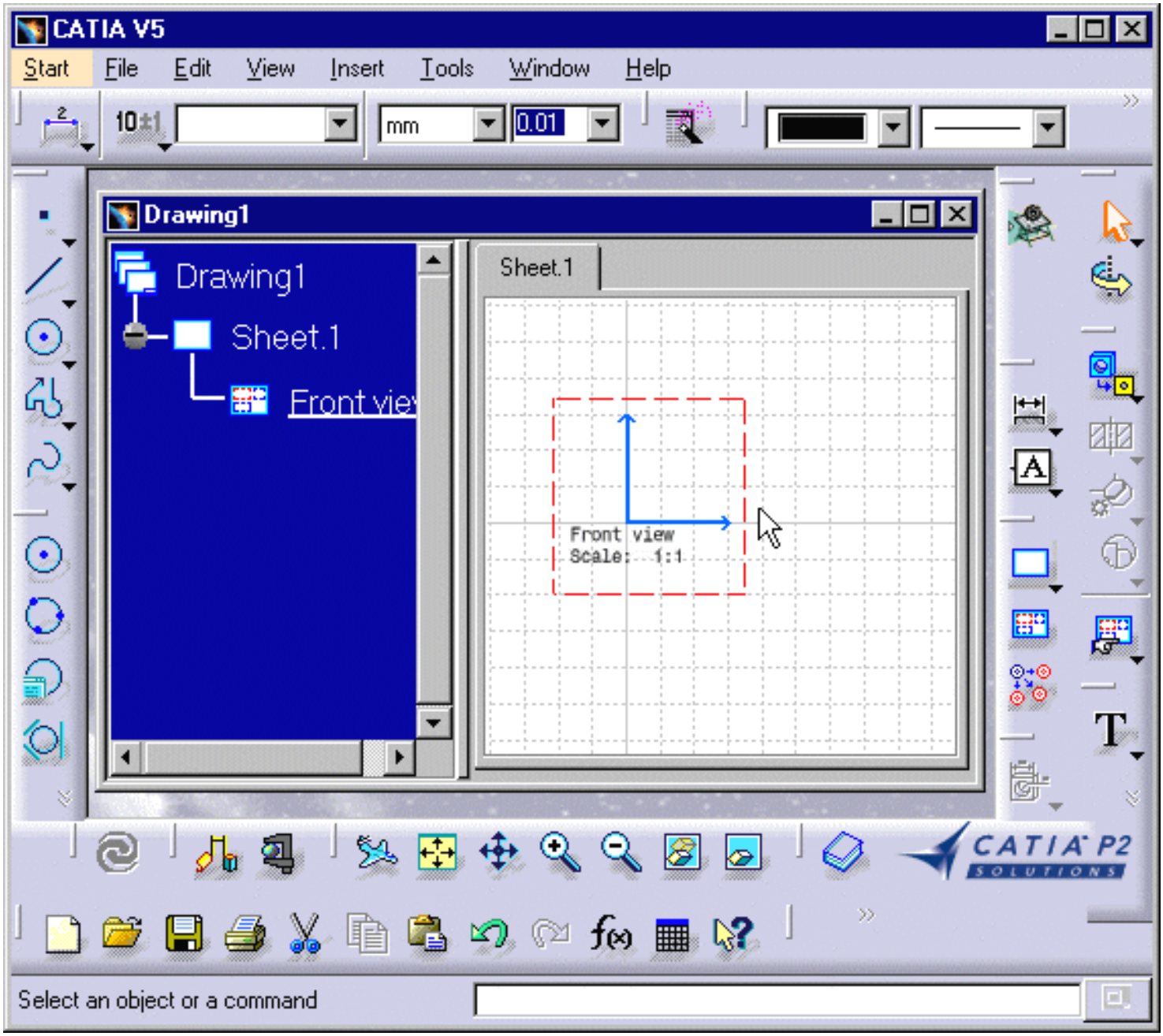
The commands for creating and editing features are available in the workbench toolbar. Now to fully discover the Interactive Drafting workbench, let's perform the following tasks.

Creating a New View

In this task you will learn how to create a new view in the empty drawing you just opened using the Drafting Interactive workbench.

1. Click the New View icon and click the Drawing sheet.

2. Click to position the new view. By default, this new view will be a front view.



In the following tasks, you will learn how to draw geometry in the empty view displayed which is by default a front view. In other words, you will draw geometry in this empty view and create both annotations and dimensions on this geometry.

Creating a Rectangle

This task shows you how to define geometry in the newly created empty view which is by default, the front view. In this particular case, let's create a rectangle.

1. Click the Rectangle icon from the Geometry creation toolbar (Profiles subtoolbar).

The Tools toolbar displays two value fields: horizontal value (H) and vertical value (V).

The Tools toolbar displays not only command options but also given fields and values that appear in accordance with the command you select. This toolbar is situated at the bottom right of screen.
If you cannot see it properly, just undock it.

2. Enter the First Point coordinates. For example, H: 0in and V: 0in.

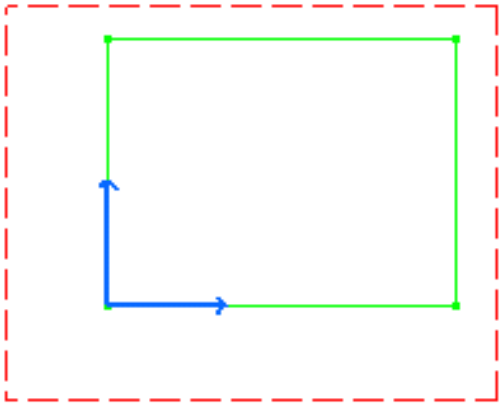
3. Press Enter.

At this step, you can either enter the rectangle second point or width and height values.

4. Enter the Second Point coordinates. For example, H: 3.5in and V: 2.5in.

5. Press Enter to end the rectangle creation.

The rectangle appears in the empty view.



You can also move the cursor for directly positioning the second point. The corresponding values similarly appear on the Tools toolbar.

Note that the grid is not necessarily displayed throughout this documentation. Still, in the **Generative Drafting** workbench, the grid is set by default. If you need to display the grid, go to `Tools->Options` (`Drafting/General`) and check the Display option.

Creating Corners

This task shows you how to create corners on an existing rectangle by multi-selecting points.

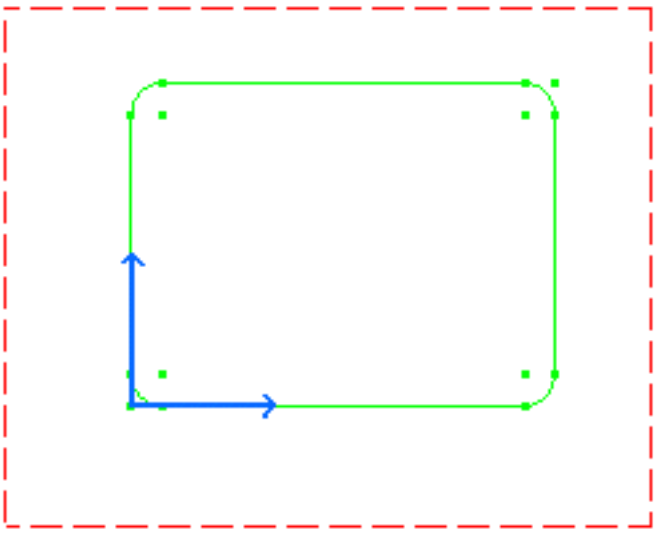
1. Multi-select the rectangle endpoints.

2. Click the Corner icon  from the Geometry Modification toolbar (Relimitations subtoolbar).

The Tools toolbar displays with a Radius field:

3. Enter a radius value in the Tools toolbar. For example, Radius: 0.25in.

4. The four corners are automatically created with the same radius value.



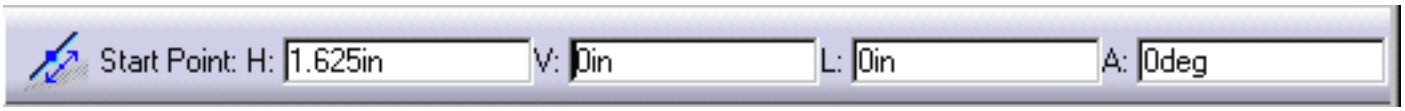
If you want to create the corners one after the others, you can also select the Corner icon first and then click the geometry.

Creating Lines

In this task you will learn how to create a line.

1. Click the Line icon  from the Geometry creation toolbar.

The `Tools` toolbar displays with the `Start Point` value fields:



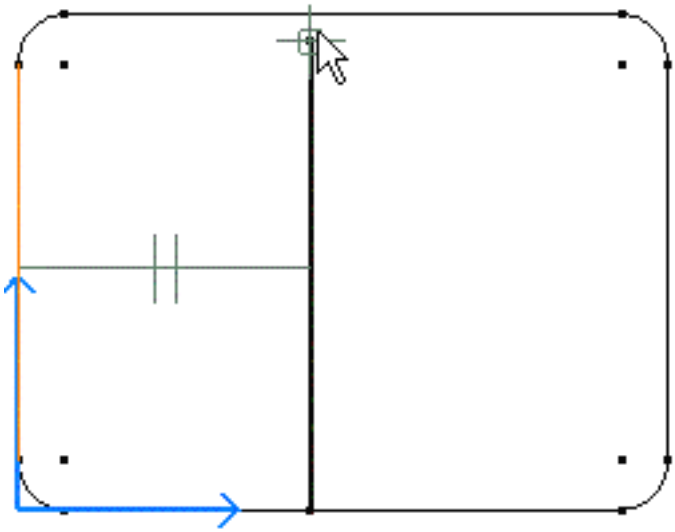
2. Enter the line `Start Point` coordinates. For example, `H: 1.625in` and `V: 0in`.

3. Press `Enter`.

4. Drag the cursor to the desired location for creating the second line point. For example, drag the line end point to the top rectangle horizontal line.

In this particular case, smartpicking is used for creating the line. In other words, you want the line to be parallel with one of the rectangle lines.

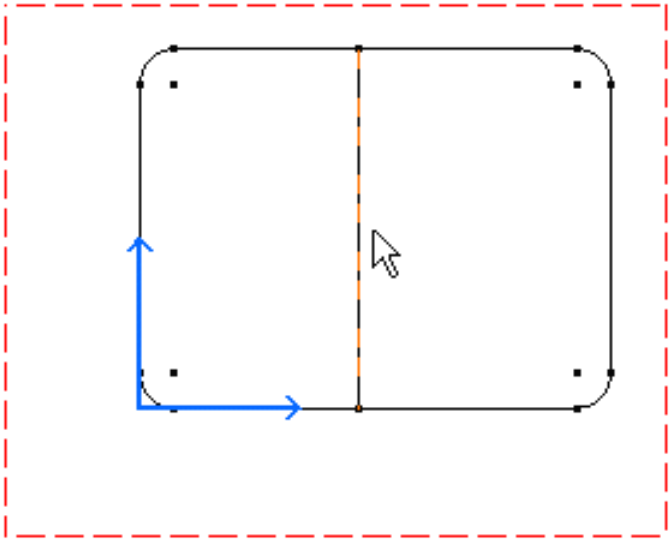
The parallelism symbol  appears as shown here.



Translating Lines

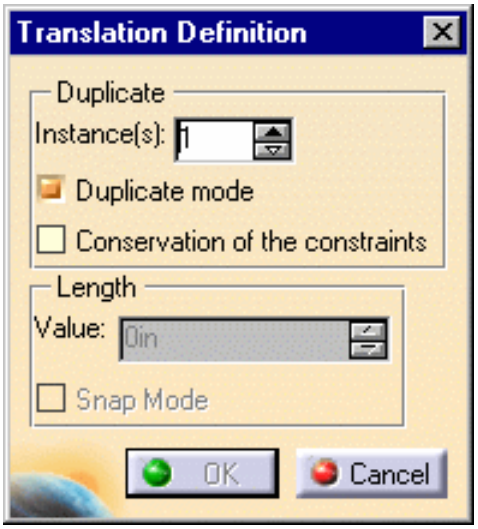
This task shows you how to translate a line. In this particular case, we will also duplicate the line to be translated.

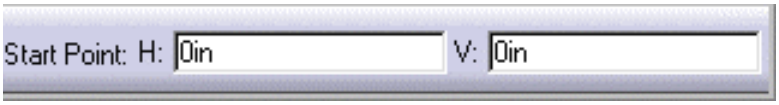
1. Select an element. For example, a line.



2. Click the Translate icon  from the Geometry Modification toolbar (Transformations subtoolbar).

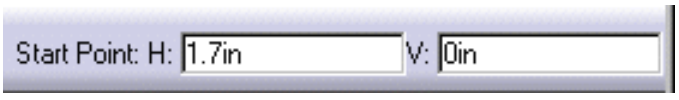
The `Translation Definition` dialog box appears and the `Start Point` value fields (H and V) appear in the `Tools` toolbar.





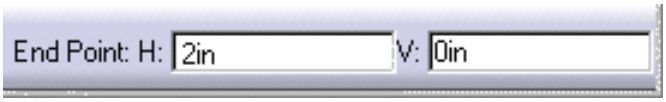
3. The `Duplicate mode` option (`Translation Definition` dialog box) is activated, by default. If not, activate this mode.

4. Enter the duplicated line `Start Point` coordinates in the Tools toolbar. For example, `H: 1.7in` and `V:0in`.



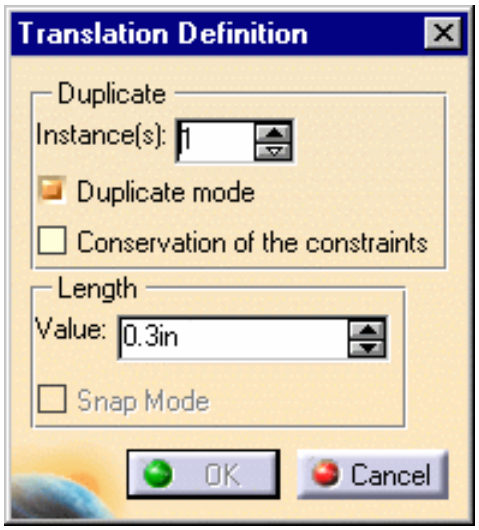
5. Press `Enter`.

6. Enter the duplicated line `End Point` coordinates in the Tools toolbar. For example, `H: 2in` and `V:0in`.



OR

5. Enter a length for the line in the `Value` field. For example, `0.3in`.

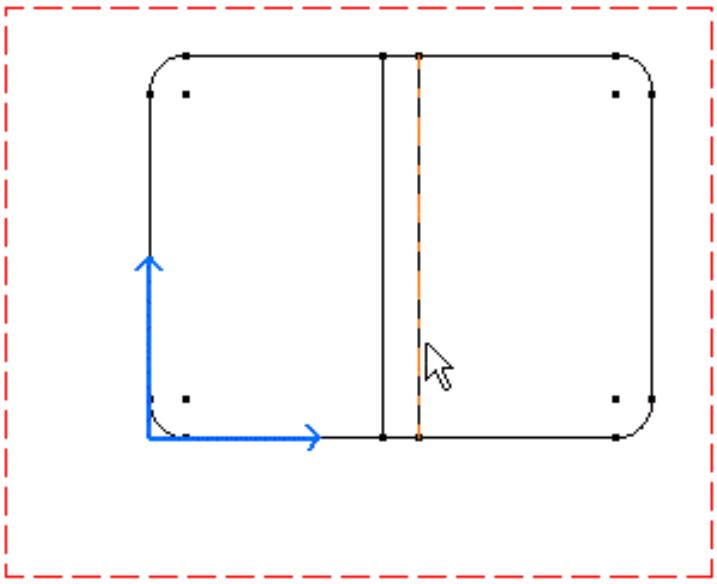


The `Snap Mode` is automatically deactivated.

6. Click `OK` to validate.

7. Once you are satisfied with your operation, click on the view.
The second line is created.

This is the resulting translated line.

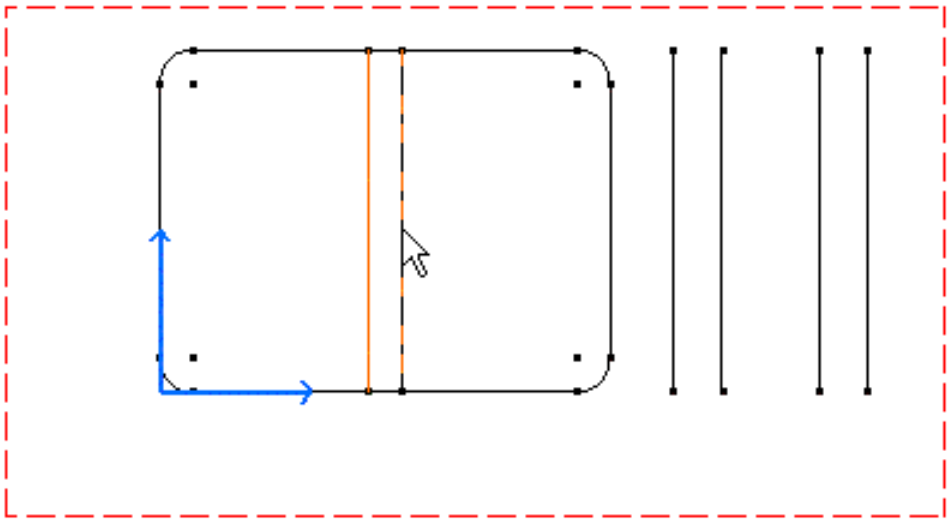


A new line is created and translated according to the existing one.

Proceed in the same manner to create the third, fourth, fifth and sixth lines. The process described above is valid for any other line to be created with the Translation command in our context.

Select two lines at a time to perform your translation, it is time-saving.

Your final drawing will look like this:



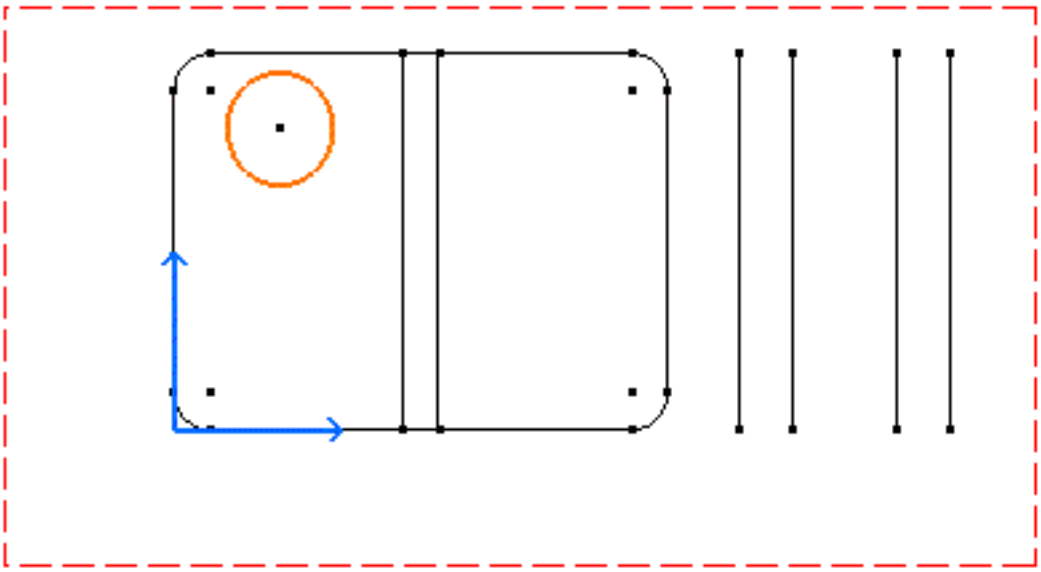
You can also select the Translate icon first and then the geometry to be translated.

Creating Circles

This task shows you how to create circles and circle centers using coordinates.

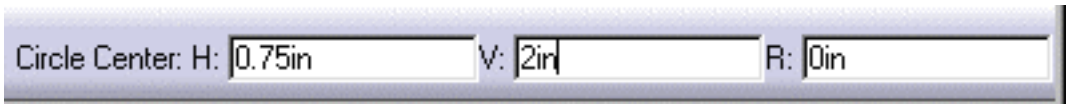
1. Select the Circle icon  from the Geometry creation toolbar.

The Tools toolbar displays circle value fields.



2. Enter the Circle Center coordinates. For example, H: 0.75in and V: 2in.

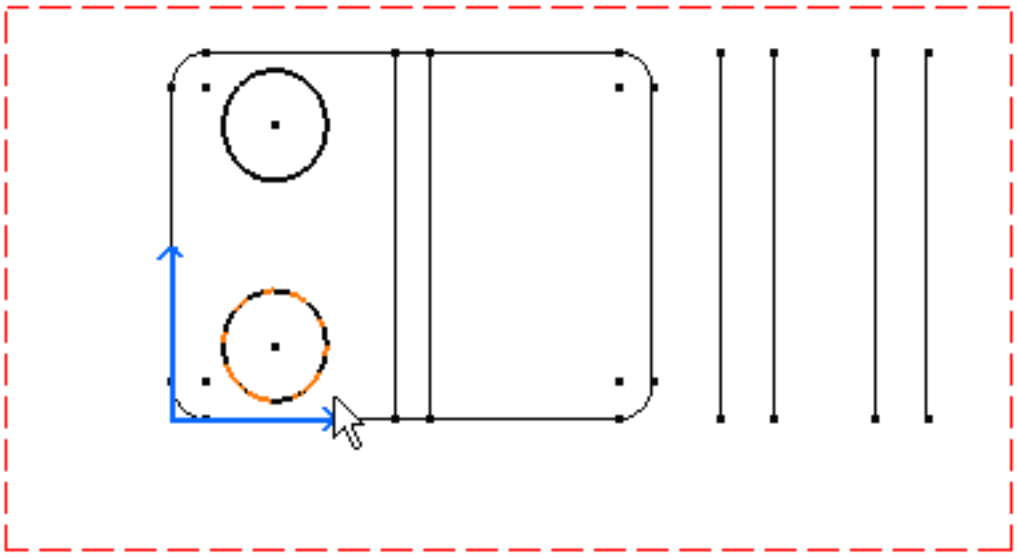
3. Press Enter.



4. Enter the circle radius. For example, R: 0.375in.

5. Press Enter.

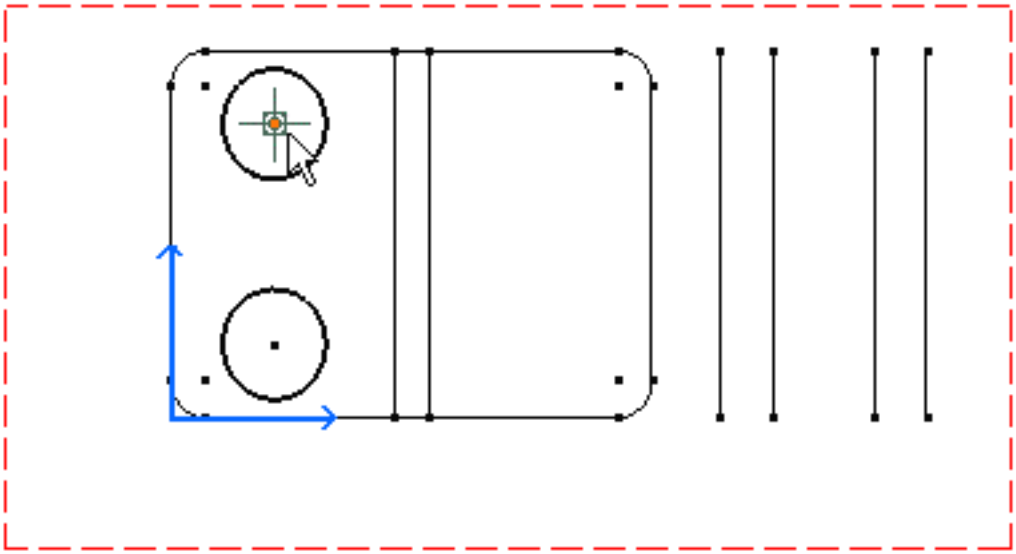
6. Repeat the scenario to create the second circle using the same circle radius values.



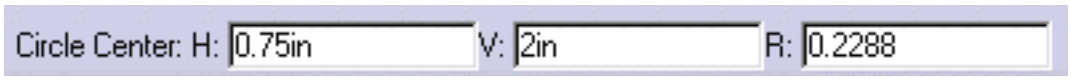
Now, let's create inner circles. For this:

7. Click again the Circle icon  .

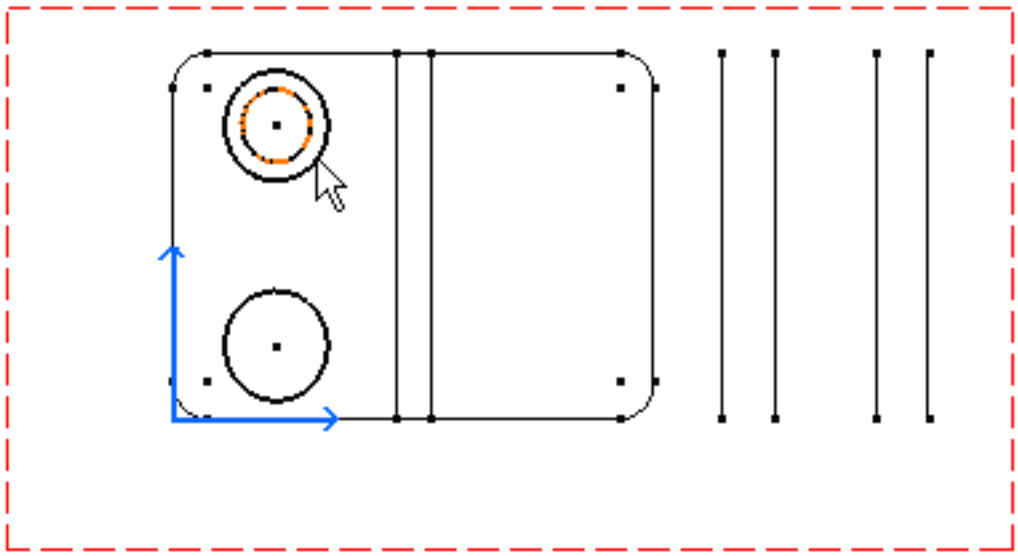
8. Select the existing circle center.



9. Enter the center circle radius.

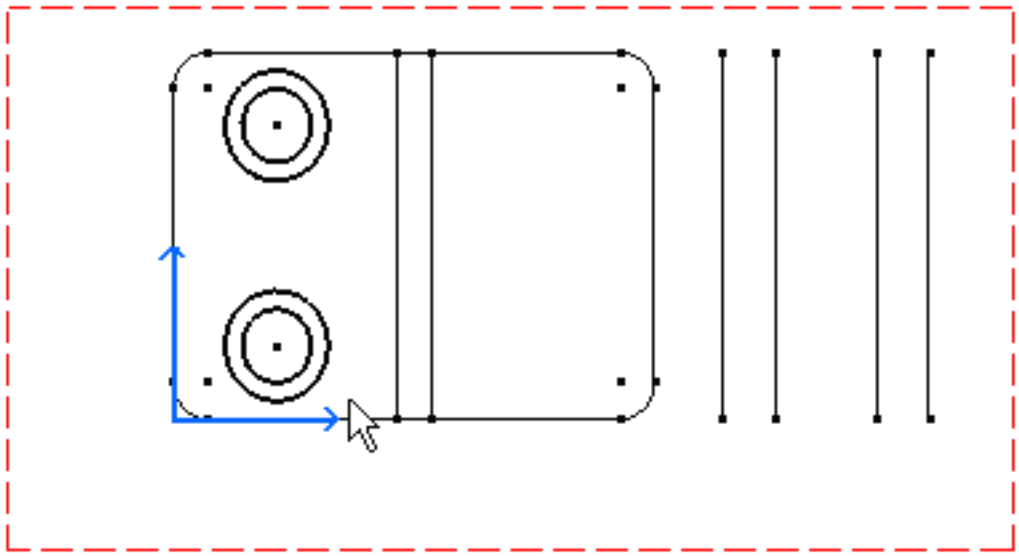


10. Press Enter.



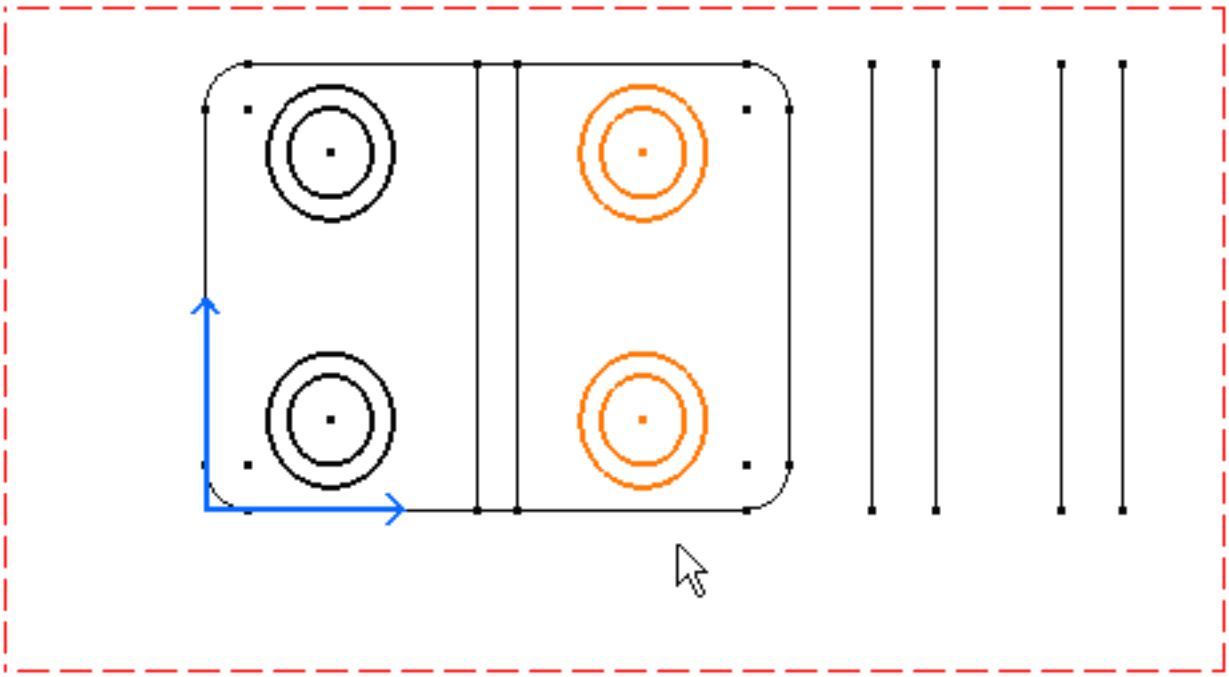
11. Repeat the scenario to create the second inner circle.

This is what you obtain:



You can also select the geometry to be translated first and then the Translate command  .

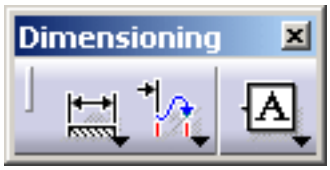
You can then translate the circles newly created and get the following result:



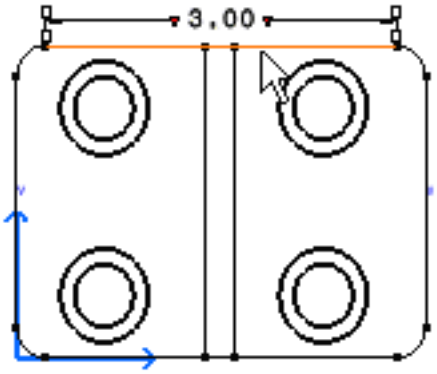
Creating Dimensions

This task shows you how to add dimensions to the geometry you previously created.

1. Click the Dimension icon  from the Dimensioning toolbar.

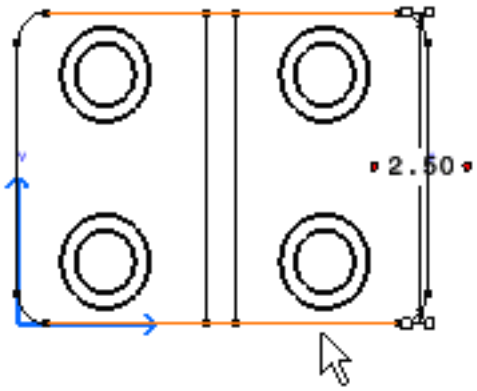


2. Click a first element in the view. For example, the rectangle top line.

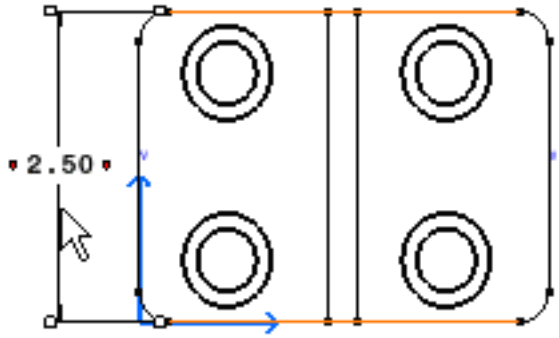


At this step, a dimension appears (length dimension). This dimension is defined according to the element first selected. You can either accept the dimension (click in the free space) or select another element (for creating a distance dimension).

3. Click a second element in the view. For example, the rectangle bottom line.



4. If needed, drag the dimension to the desired location.



At this step, you can apply various modifications to the dimension you are creating. You can:

- modify the dimension overrun/blanking using manipulators or the Ctrl key to modify only one extension line.
- add text before or after by double-clicking the dimension
- redefine the dimension properties using the required toolbar:

Creating Annotations

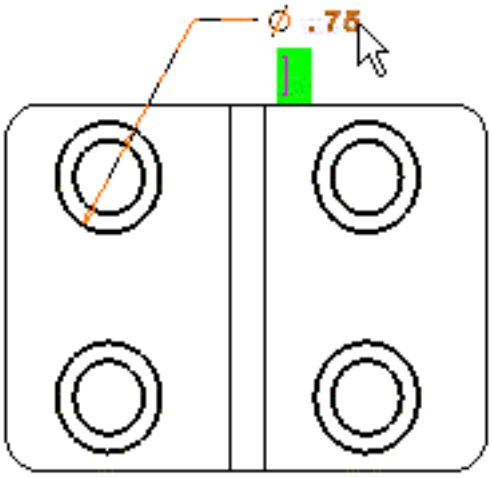
This task shows you how to add annotations on your drawing. In this particular case, we will add text to existing 2D elements.

1. Click on an icon from the Annotations toolbar.

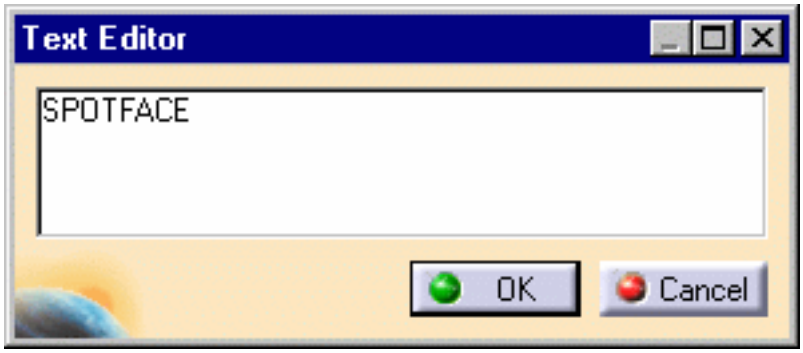
For example, click the Text icon  .

2. Click an element.

The text will be positioned according to this element.

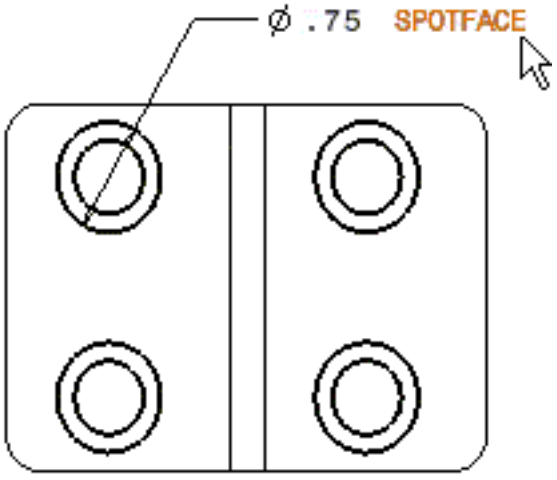


3. Enter the required text in the Text Editor dialog box.



As you type in, the text appears in the graphic Text Editor window.

4. If needed, drag the text to the desired location.



The annotation will now remain associated to the selected 2D element. In other words, each time you move the 2D element, the associated annotation moves accordingly.