



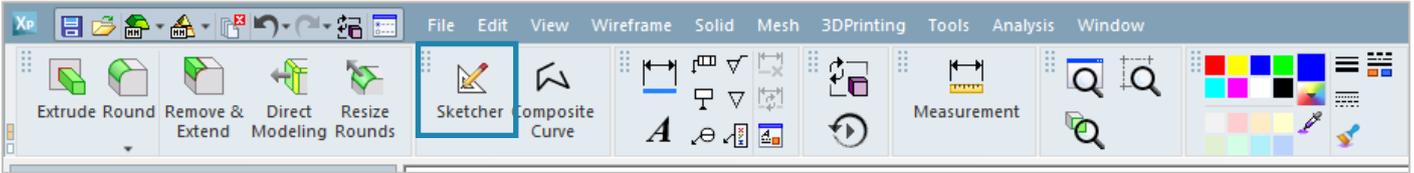
3DXpert™ for SOLIDWORKS®

Part Design

Sketcher - Basic 1

Tutorial V3: 14,0200,1606,1028(SP2)

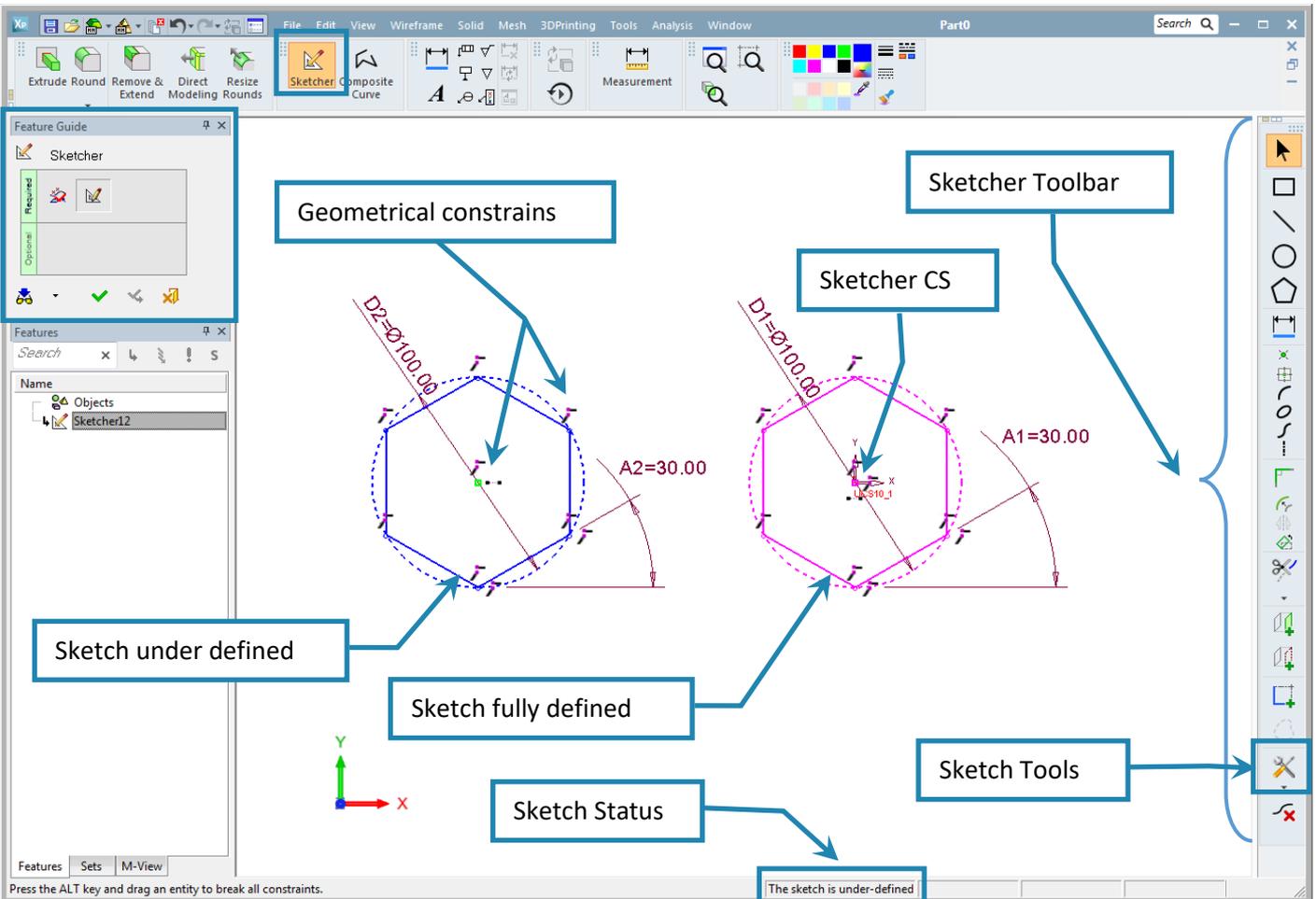
In this exercise, we will learn the foundation of the **Sketcher** and its basic functions.



The **Sketcher** is a tool used to create two-dimensional (2D) geometries on plains only, including on the plains of the part.

These Sketches are a basic building stone for most of the actions in creating a solid bodies as well as they can be used as curves on their own for different usage.

Within the **Sketcher**, the user can create lines, circles, arcs, polygons, dots, ellipses, and more. He can give dimensions, add geometrical constrains, trim and/or extend entities, deals with corners, chamfers and rounds. The user can also use tools for sketching with symmetry, for copy and move entities and more. The user can add, remove and edit dimensions and geometrical constrains easily at any time and create parametric relations.



| | | |
|-------------------------------|--|--|
| <p>! Notice/ Remember</p> | | Left mouse button name is " <i>pick</i> " |
| | | Middle mouse button name is " <i>Exit</i> " |

Sketcher – general information

Sketch plane

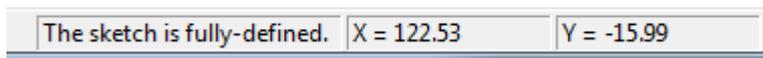
After **picking** the **Sketcher** command it is necessary to choose the sketch plane. The following options are available:

- By pressing the **exit**, we can use the system default to open the sketch plane on main XY plain.
- **Picking** 2 of the 3 major axis XY, YZ, XZ.
- **Picking** any planar face of the part.
- **Picking** geometries that define a plane, i.e. 3 points or line and point.

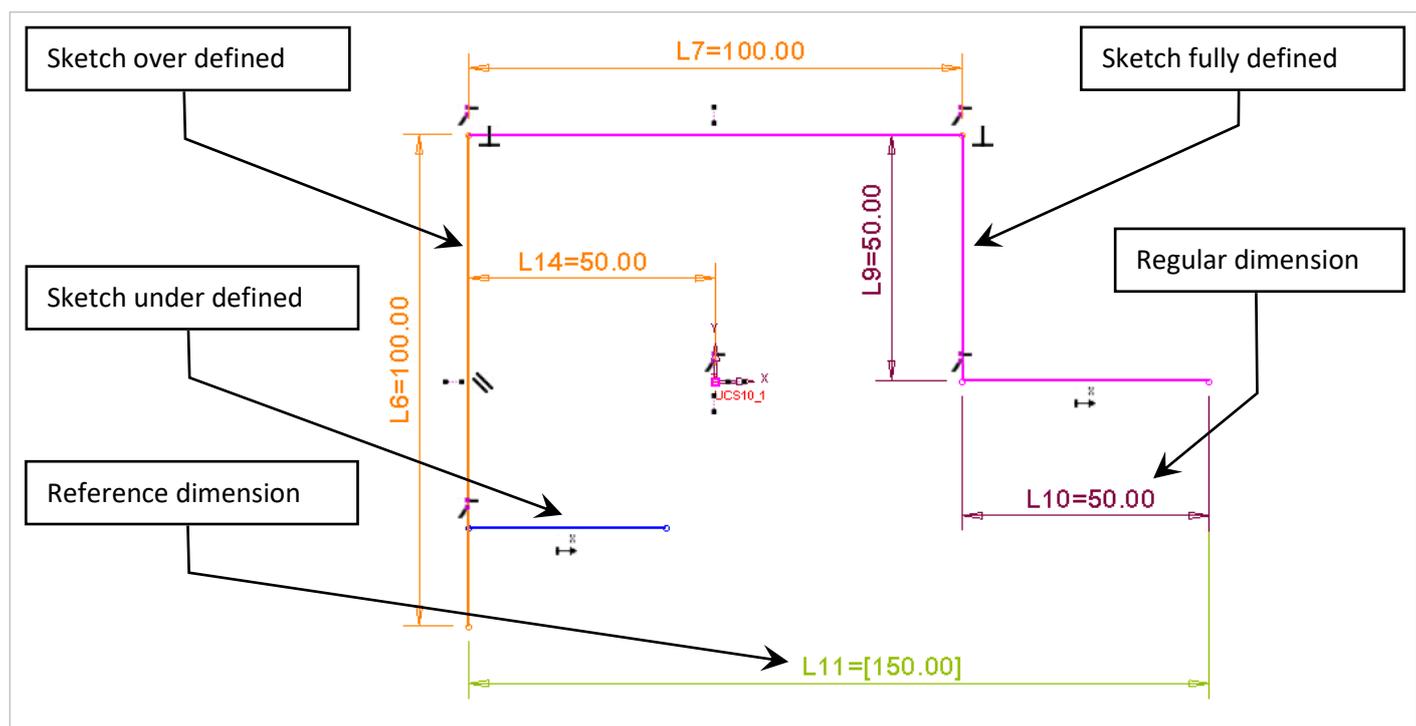
Sketcher status and colors

Sketcher status intended to show the user the progress from under defined sketch to a fully defined sketch.

The system use different colors to show the status of the sketch as well as having a status bar at the bottom of the screen:



| | |
|--|--|
| | Blue color mean that the sketch is under defend Add dimensions and geometrical constrains until reaching fully defend sketch. |
| | Pink color mean that the sketch is fully defend This is the desired and recommended situation – time to "OK" <input checked="" type="checkbox"/> . |
| | Orange color mean that the sketch is over defend Do not continue – check why got it over defend and fix it. |
| | Red means that there is inconsistency Do not continue – check why got it inconsistent and fix it. |
| | Purple is for geometries that came from "Add reference" and/or "Add geometry" The sketch is fully defend - No need to change the color. |
| | Green related to a reference dimension. |



Applicable sketcher for solid

Applicable sketch for solid is basic a sketch that is not intersect herself and it can close or open (open sketch for "thin wall on" mode in some solid functions).

| | | | |
|-----------------------|--|--|--|
| Applicable sketch | | | |
| Not applicable sketch | | | |

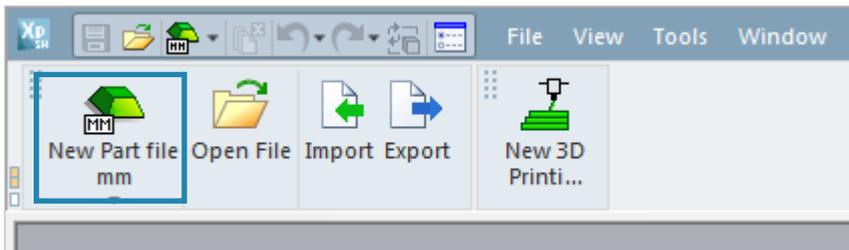
! Please notice: A Solid object will be generated based on a valid sketch only, even if the sketch is partially defined.
For the best practice it is recommended to work with pink - "fully defined" sketches.

Sketcher Toolbar

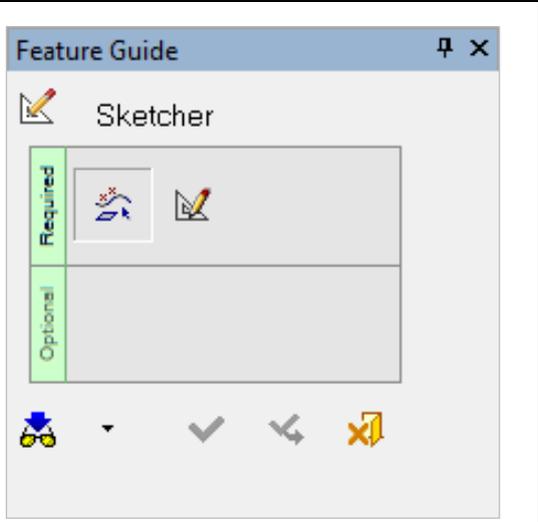
| | | |
|--|--------------------|--|
| | Select | Pick entities in the current sketch. It is also useable to "drag" and move undefined points, lines and arcs. |
| | Rectangle | Use to build a rectangle |
| | Line | Use to build a line |
| | Circle | Use to build a circle |
| | Polygon | Use to build a polygon |
| | Dimension | Use to entities |
| | Point | Use to build a point |
| | Center Of Geometry | Create a point or points in the center of <i>picked</i> entities |
| | Arc | Use to build an arc |

| | | |
|---|-----------------------|--|
|  | Ellipse | Use to build an ellipse |
|  | Spline | Use to build a spline |
|  | Symmetry | Sketch entities under symmetry condition |
|  | Corner | Create corners, chamfers and rounds |
|  | Offset | Create offset to entities |
|  | Mirror | Create copy mirror Through line |
|  | Copy/Move/Rotate | Copy, move or rotate picked entities |
|  | Trim Tools | Changeable icon according to <i>picked</i> trim function selected |
|  | Trim | Trim overlapped who intersect each other |
|  | Trim (Split) / Extend | Trim, split or extend entities |
|  | Add Geometry | Pick lines, circles, points etc. from the model to be added as geometry of the sketch |
|  | Add Reference | Pick lines, circles, points etc. from the model to be use as reference for the sketch |
|  | Add Constraints | Add geometrical constraints to entities (to save dimensioning) |
|  | Turn To Reference | Pick entities to turn them to reference and vice versa |
|  | Sketch Tools | Additional Sketcher commands |
|  | Delete | Use to delete <i>picked</i> entities and dimensions |

1. From the main menu **pick** "New Part File":

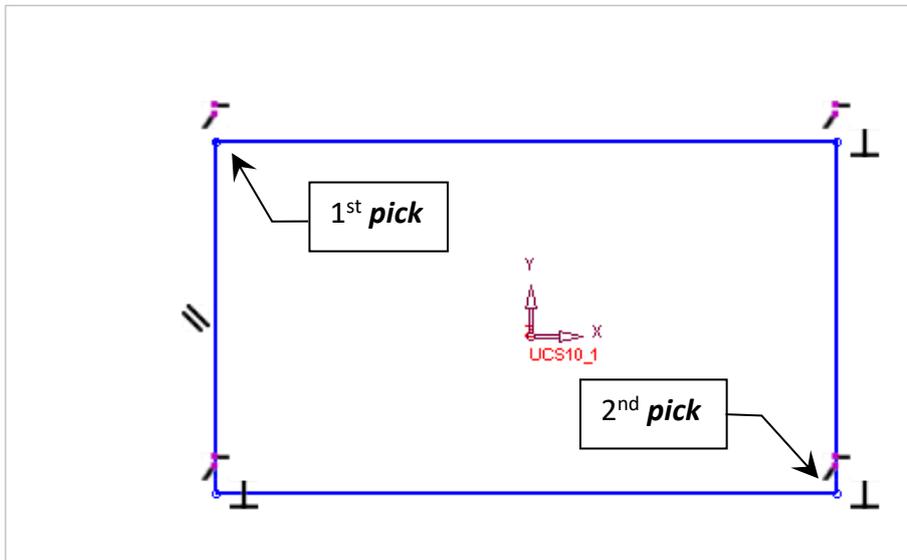


2. From the toolbar **pick** the "Sketcher" Command  ,

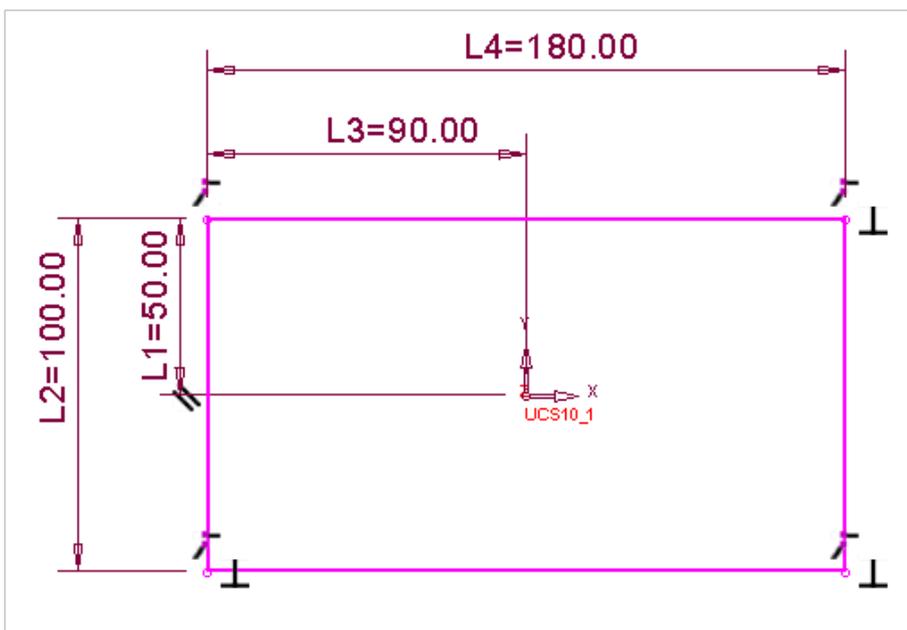
| | | |
|---|---|---|
|  | Sketcher |  |
|  | pick entities to define the plan | |
|  | Sketch | |
| | | |
|  | To approve and finish use the " OK " | |
|  | To approve and continue use the " Apply " | |
|  | "Cancel" – exit the command without keep changes | |

Press the **Exit** (middle mouse button), the system default will open the sketch plane on main XY plain.

3. **Pick** the rectangular Command  and **pick** 2 points on the screen as shown:



Give dimensions, notice the order of dimensioning (L1, L2, L3, L4) until it is **Fully Defined**:



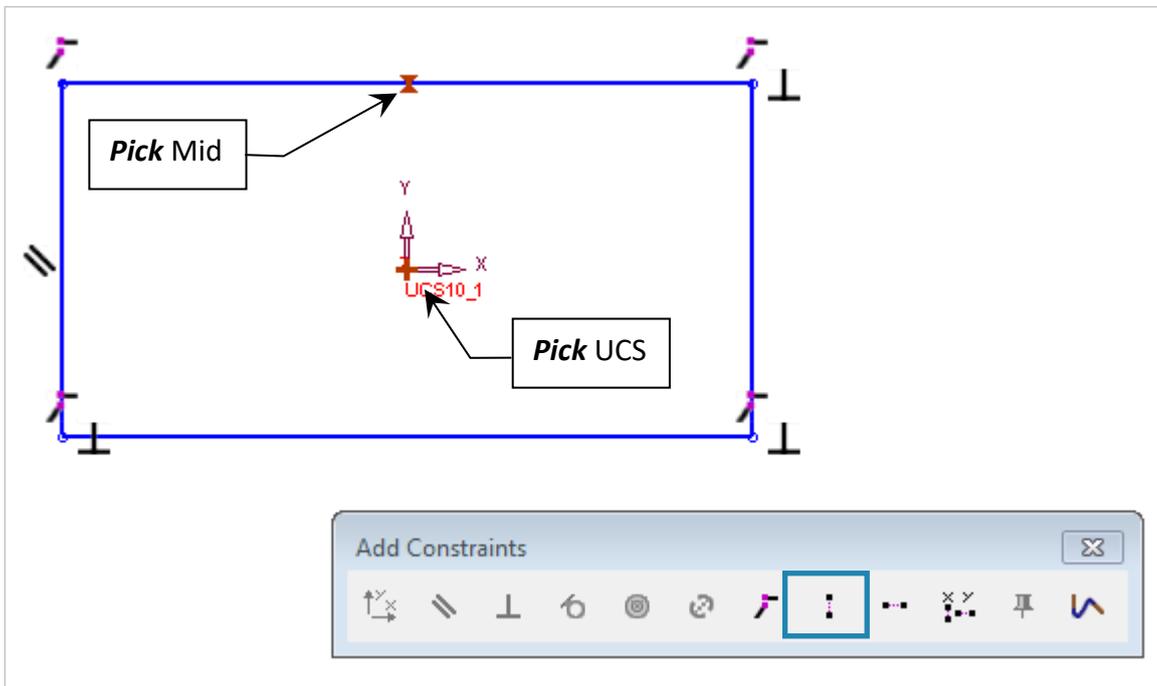
In the following exercises we will do several sketches until they are pink "Fully Defined".

For time saving instead of opening a new file each time it is an option to "Cancel Sketch" and immediately start the next exercise.



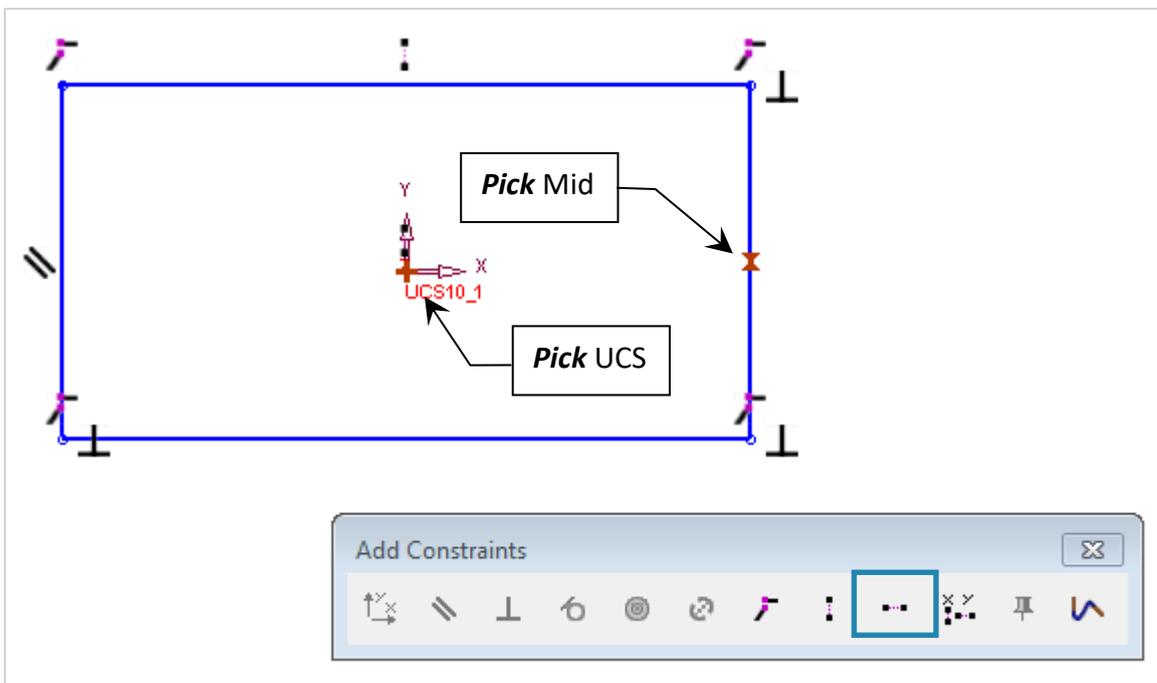
4. Start a new sketch, **pick** the Rectangular command , **pick** 2 points to create it.

5. **Pick** the Add Constraints command  and pick 2 points as shown – mid point (X) and ucs point:



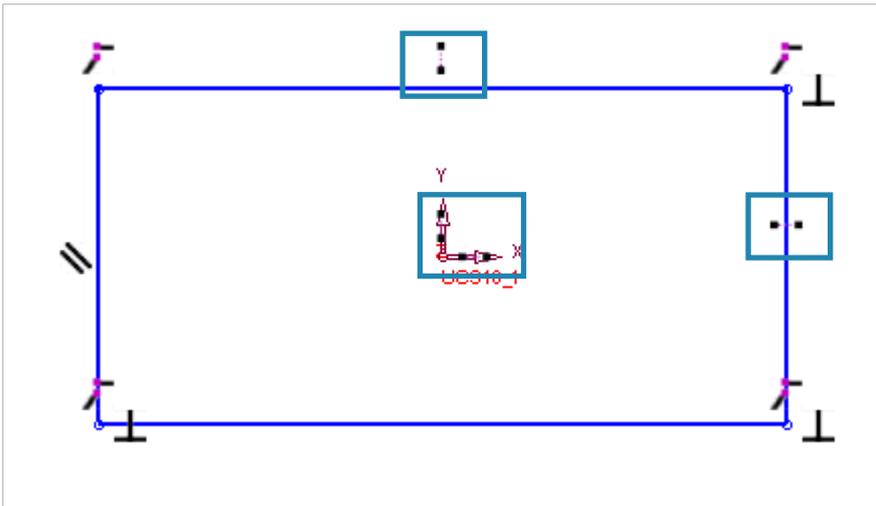
Give this 2 points the Constraint "Same X" , only the midpoint will move to align with the UCS.

6. Repeat and pick 2 points as shown – mid point (Y) and UCS point:



Give this 2 points the Constraint "Same Y" , only the midpoint will move to align with the UCS.

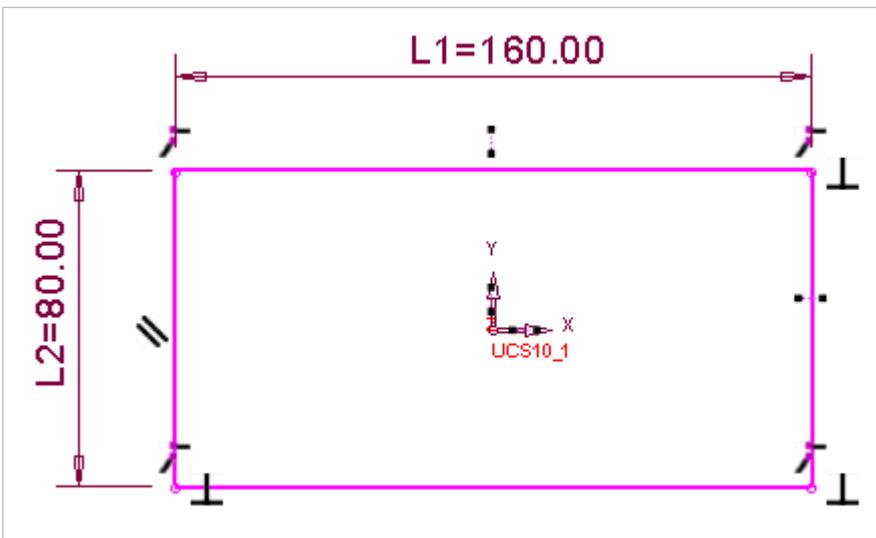
It possible to see on the sketch the geometric constraints, note that there is 2 pairs of them.



Close the Add Constraints menu.

! By using the select , it is possible to drag and move lines or points. The movement will be identical for both sides since the mid points are locked. So it will be when giving dimensions.

7. Give dimensions as shown:



End of Exercise.