Sketcher Enhancements

Introduction

Wildfire 4.0 includes several enhancements to the sketcher as listed below.

Auto Diameter Dimensions When creating sketches for revolved features, the system will now create diameter dimensions across the centerline as sketch entities are created. The centerline must be created before the sketch entities. If the sketch has multiple centerlines, the dimensions are created using the first centerline added to the sketch. Set the following new configuration option to enable this behavior. `sketcher_dim_of_revolve_axis yes`

Sketcher Colors The color scheme used in the sketcher can now be set using the **System Colors** dialog box.

Sketcher Diagnostic Tools The new **Sketcher Diagnostic Tools** allow you to verify the integrity of the sketch before picking the checkmark to complete the sketch.

Sketch Line Style The **Line Style** of sketch entities (lines, arcs, circles, splines, and text) can now be changed using the **Line Style** dialog box.

Sketch References The **References** dialog box can be used to ***replace*** missing or invalid references in the sketcher, and you can now use **Intent Objects** as sketch references.

Undo View Orientation When you accidentally spin the sketch view, you can now use **Undo** to return to the sketch view. Set the following new configuration option to enable this new behavior.

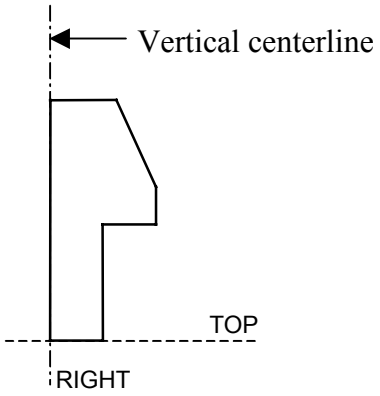
`sketcher_undo_reorient_view yes`

Tutorial – Auto Diameter and Undo Spin

- Pick **Tools, Options**
- **Add** and **Apply** the two configuration options listed below:

```
sketcher_dim_of_revolve_axis  yes
sketcher_undo_reorient_view  yes
```

- **Close** the **Options** dialog box
- Create a **New** part called < revolve_me >
- Pick the **Revolve** icon then press and hold the right mouse button and select **Define Internal Sketch**
- Select the FRONT datum plane then pick **Sketch** in the dialog box
- Sketch a vertical centerline aligned with the RIGHT datum plane
- Sketch the geometry as shown below

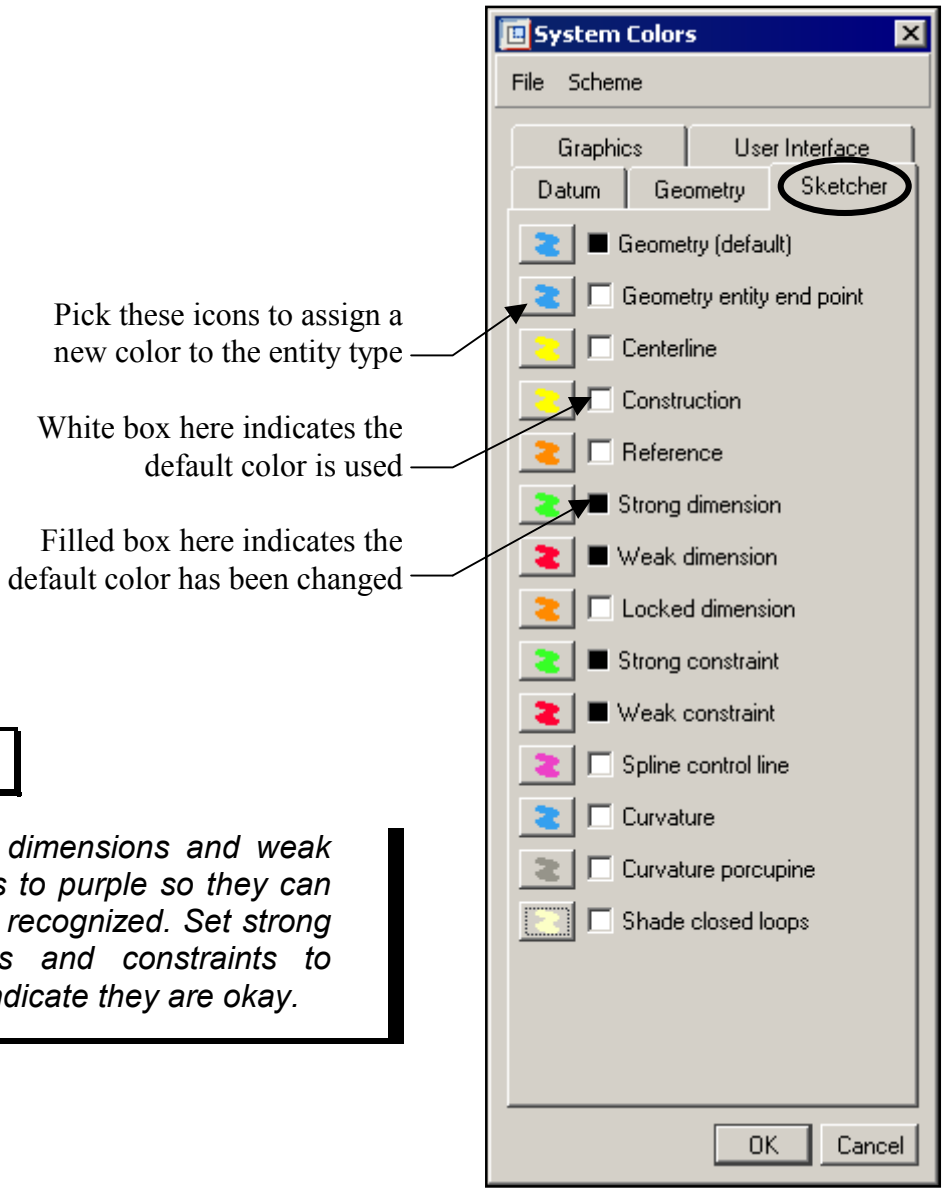


- After sketching the geometry, notice the system automatically creates diameter dimensions across the centerline
- Spin the view then pick the **Undo** icon
- Pick the checkmark to complete the sketch
- Pick the checkmark in the dashboard tool to complete the revolve feature
- **Save** the part and **Close** the window
- The **Save Object** dialog box is used with the **Save** command because this is the first time this object has been saved

Sketcher Colors

The color scheme used in the sketcher can now be set using the **System Colors** dialog box.

- Pick **View, Display Settings, System Colors**.
- In the **System Colors** dialog box, pick the new ***Sketcher*** tab then pick the appropriate icon as shown below.
- Colors that have been changed from the default are indicated with a filled box as shown below.

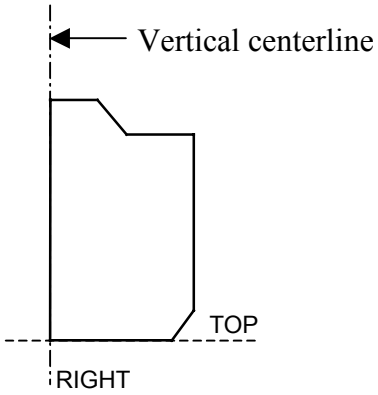


Tip

Set weak dimensions and weak constraints to purple so they can be quickly recognized. Set strong dimensions and constraints to green to indicate they are okay.

Tutorial – Sketcher Colors

- Create a **New** part called < sketch_colors >
- Pick **View, Display Settings, System Colors**
- In the **System Colors** dialog box, pick the new ***Sketcher*** tab
- Pick the color icon next to **Weak Dimension** then pick a purple color
- Pick the color icon next to **Weak Constraint** then pick a purple color
- Pick the color icon next to **Strong Dimension** then pick a green color
- Pick the color icon next to **Strong Constraint** then pick a green color
- Pick **OK** in the **System Colors** dialog box
- Pick the **Revolve** icon  then press and hold the right mouse button and select **Define Internal Sketch**
- Select the FRONT datum plane then pick **Sketch** in the dialog box
- Sketch the geometry as shown below

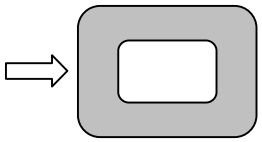


- After sketching the geometry, notice the new color scheme you just set
- Pick a weak dimension then pick **Strong** in the right mouse button popup menu
- Pick a weak constraint then pick **Strong** in the right mouse button popup menu
- Pick the checkmarks to complete the sketch and the revolve feature
- **Save** the part and **Close** the window

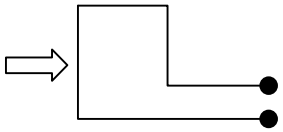
Sketcher Diagnostic Tools

The **Sketcher Diagnostic Tools** is a collection of four new commands as listed below.

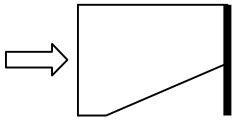
Shade Closed Loops Shade the inside of closed loops in the sketch. This is a great tool for analyzing the sketch, especially for beginner users.



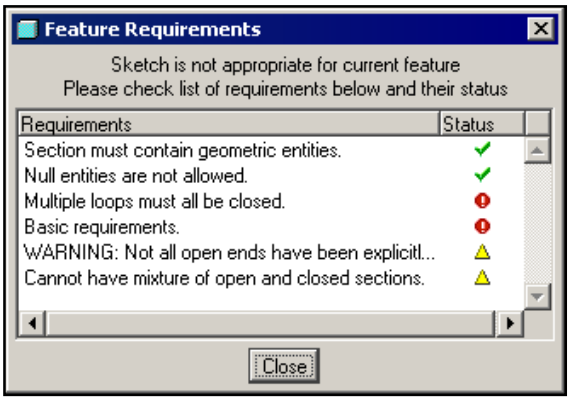
Highlight Open Ends Highlight the ends of entities that are not connected to other entities in the sketch. This is another great tool for finding errors in the sketch.



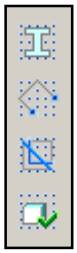
Overlapping Geometry Highlight entities that overlap. The entire entity is highlighted, not just the portion that overlaps.



Analyze Sketch Verify the integrity of the sketch and open the **Feature Requirements** list as shown below.



The **Sketcher Diagnostic Tools** toolbar is shown below.



Shade Closed Loops

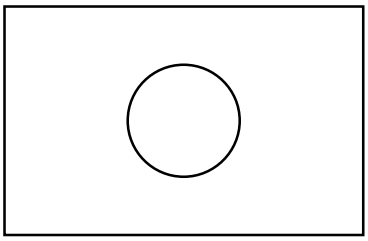
Highlight Open Ends

Overlapping Geometry

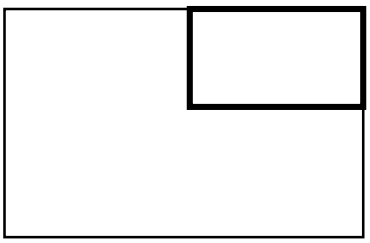
Analyze Sketch

Tutorial – Sketcher Diagnostic Tools

- Create a **New** part called < sketch_diagnostics >
- Pick the **Extrude** icon  then press and hold the right mouse button and select **Define Internal Sketch**
- Select the FRONT datum plane then pick **Sketch** in the dialog box
- Sketch a circle inside a rectangle as shown below

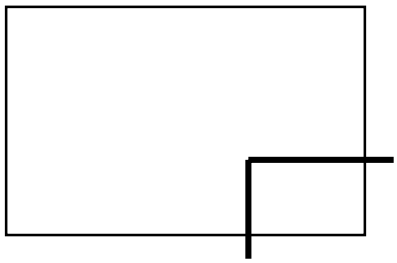


- Pick the **Shade Closed Loops** icon 
- Pick the **Analyze Sketch** icon 
- Read the information in the **Feature Requirements** list then pick **Close**
- Delete the circle then add another rectangle as shown below in bold

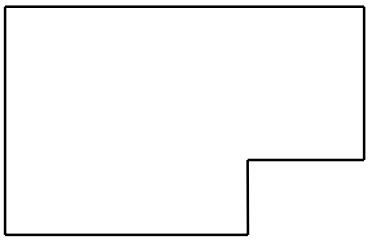


- Pick the **Overlapping Geometry** icon 
- Notice the system highlight the entities that overlap
- Pick the **Analyze Sketch** icon 
- Read the information in the **Feature Requirements** list then pick **Close**
- Pick the **Undo** icon  to delete the second rectangle

- Sketch two lines, extending past the original rectangle as shown below in bold



- Pick the **Highlight Open Ends** icon 
- Notice the system highlights the endpoints of the two lines
- Pick the **Analyze Sketch** icon 
- Read the information in the **Feature Requirements** list then pick **Close**
- Trim the sketch as shown below



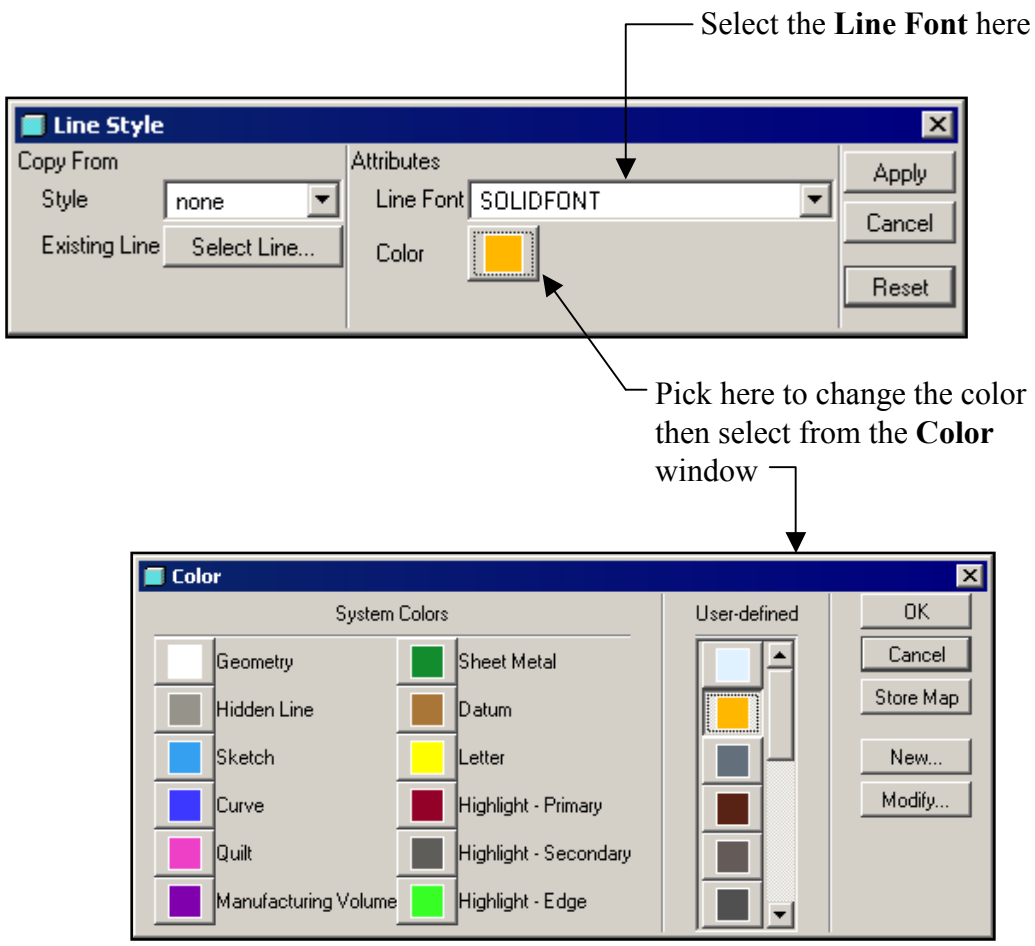
- Use the four new diagnostic tools to be sure the sketch is correct
- Pick the checkmark to complete the sketch 
- Pick the checkmark in the dashboard tool to complete the extrude feature
- **Save** the part and **Close** the window

Sketcher Line Style

Sketches entities can now have alternate line styles and colors applied.

- For new entities, pick **Sketch, Line Style, Set Line Style** before creating the sketch.
- To reset the sketch style to the default, pick **Sketch, Line Style, Clear Line Style**.
- To change the style of existing sketch entities, select the sketch entities first then pick **Properties** in the right mouse button popup menu.
- The **Line Style** dialog box is used to select the **Line Font** and **Color** as shown below.
- Geometry can be imported into the sketcher with its original line style using the following new configuration option:

```
sket_import_geometry_line_style  yes
```



Tutorial – Sketch Line Style

- **Open** the part called 'sketch_style.prt'
- Pick the **Sketch 1** feature then pick **Edit Definition** in the right mouse button popup menu
- Select all the lines in the sketch then pick **Properties** in the right mouse button popup menu
- Change the **Line Font** and the **Color** as appropriate
- Pick **Apply** and **Close** in the **Line Style** dialog box
- Pick **Sketch, Line Style, Set Line Style**
- Change the **Line Font** and the **Color** as appropriate (this is for new entities)
- Sketch a circle inside the part
- Use the four new diagnostic tools to be sure the sketch is correct

Shade Closed Loops
Highlight Open Ends
Overlapping Geometry
Analyze Sketch

- Pick the checkmark to complete the sketch 
- **Save** the part and **Close** the window

Sketch References

Features that fail because of invalid or missing sketch references can be fixed using the **References** dialog box in the sketcher.

- Missing and other failed references can now be replaced.
- This is similar to using the **Edit References** command to replace the references of the feature so that all references from other features are preserved.
- Select the reference then pick **Replace** as shown below. Select a new model entity to replace the missing reference.
- Pick **Solve** to re-evaluate the sketch references.