

## **MOST OFTEN USED SPINFIRE 8.2 FUNCTIONS**

[\*\*Window Functions\*\*](#)

[\*\*Assembly Functions\*\*](#)

[\*\*Part Manipulation\*\*](#)

[\*\*Dimension Functions\*\*](#)

[\*New in 8.2\*](#)

[\*\*Note Functions\*\*](#)

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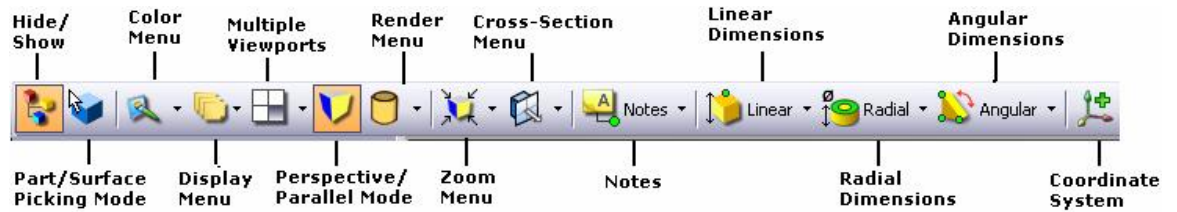
[\*\*Coordinate Systems\*\*](#)

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[\*New in 8.2\*](#)



## I. Window Functions

### Zoom Fit



The **Zoom Fit** button on the Viewer toolbar pans and zooms to provide a close-up view of the model while still fitting the model entirely within the Viewport.

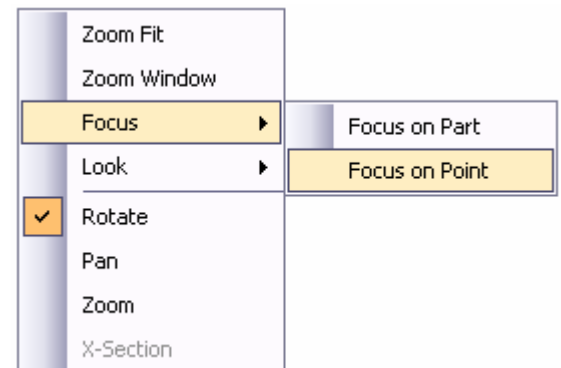
The Zoom Fit command may also be called by *right-clicking* the background in the 3D scene and selecting Zoom Fit from the menu.

### Focus Point

The **Focus on Point** command shifts the center of the view to a selected vertex point on the model. The center of the view is also the center of rotation.

#### To focus on a point:

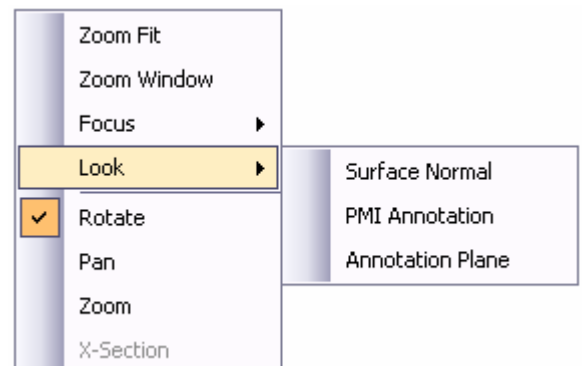
1. Open the Focus menu on the Viewer toolbar
  - o *Click* the drop-down arrow to the right of the **Focus** button. You should see a drop-down menu listing the available commands.
2. *Click* "Focus on Point".  
Red dots designate available vertices on the model. A cross-hatch indicates when the pointer is properly positioned over a vertex point.
3. *Click* a vertex on the model.  
The vertex point will shift to the center of the view.

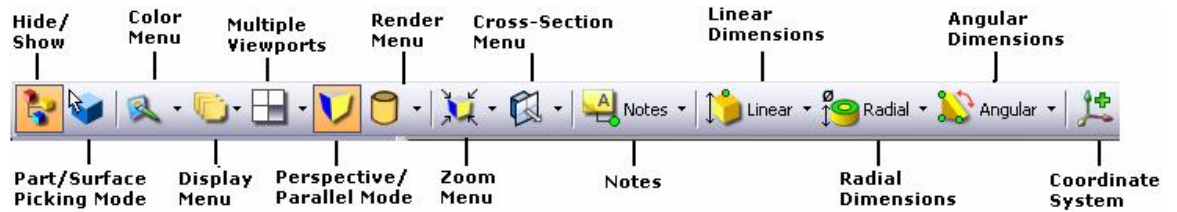


**New For 8.0**

### Look

The **Look** command allows you to select one of (3) options to obtain a different view;





To Look at **Surface Normal / PMI Annotation / Annotation Plane:**

1. Open the *Zoom Fit* menu on the Viewer toolbar
  - o *Click* the drop-down arrow to the right of the **Look** button. You should see a drop-down menu listing the available commands.
2. *Click* one of the options above.  
The Model will reposition based on the selection.

### Split Screen Viewport

- While a Section Cut Plane is defined, select the “*Viewports*” Menu and select “2 *Horizontal Viewports*” or “2 *Vertical Viewports*”

## II. *Assembly Functions*

### Show/Hide Parts and Sub Assemblies

From the **Assembly Tree** you can make any part or subassembly in a model visible or invisible.

**To Hide or Show an object in an assembly:**

1. *Click* the *Model tab* in Viewer.
2. In the **Assembly Tree**, clear or select the check box beside the object.  
A visible object will have its check box selected.

**Note:** You may also hide a part by *right-clicking* it in the 3D scene and selecting *Hide*.



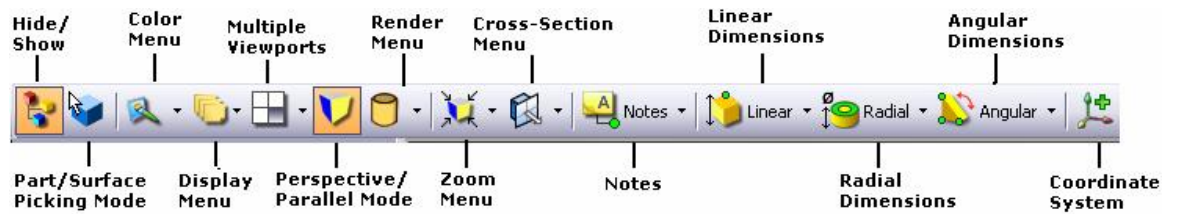
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### Create new Assembly

You may add a new assembly node to the tree. Parts and subassemblies may then be dragged to and from this node.

**To add an assembly node:**

1. Select the *Assembly* tab in the Viewer component.



2. **Right-click** the assembly or subassembly under which you would like to create the node.
  - This opens the Assembly context menu.
3. Select the **New Assembly Icon** .
  - This will add the node. The name of the node will be highlighted and the cursor positioned for you to enter the new name.
4. Type the name of the node.
5. Press **Enter**.

**Note:** You must *press* the **Enter** key for the new name to be saved. If you click the mouse instead, any changes made to the name will not be saved and the object will revert to the default name.

### **III. Part Manipulation**

#### **Add Transparency**

You can adjust the level of transparency of any part, subassembly, or assembly.

#### **To change the transparency of an object:**

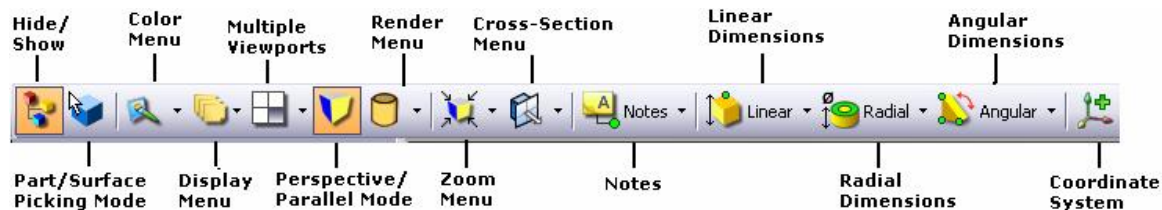
1. **Click** the **Model tab** in Viewer.
2. **Right-click** the name of the object in the **Assembly Tree**.  
This opens the Object context menu.
3. **Click Transparency**.  
This opens the Set Transparency slider.
4. Adjust the slider until the object is at the desired transparency.  
To guide you, the transparency of the object in the Viewport will reflect the real-time adjustments you make with the slider.
5. **Click** Close.

#### **Change Color**

You can determine the color for any part, subassembly, or assembly. You can choose a predefined color or define your own color. You can choose to have the color applied to the "Front" (Exterior), "Back" (Interior), or both.

#### **To change the color of an object:**

1. Click the Model tab in Viewer.
2. **Right-click** the name of the object in the assembly tree.  
This opens the Assembly Object context menu.



3. Point to Color.
4. **Click** either "Front and Back", "Front", or "Back"  
This opens the Color dialog box.
5. **Click** a color swatch. Or **click** "Define Custom Colors" to create your own color.
6. **Click** OK.

#### IV. Dimension Functions

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##### Note

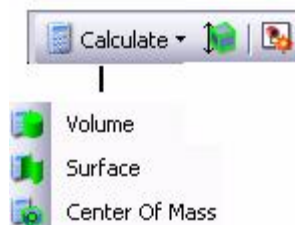
- If you want to add several of the same type of *markup*, you may select the **Lock tool** check box that appears when you select a markup command. This will allow you to keep adding the same type of markup to the scene until you click **Stop**.

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##### Part Volume

##### **To measure the Volume of a part:**


1. Select the **Assembly** pane.
2. Select the part, assembly, or sub-assembly that you would like to measure.
3. Open the **Calculate** menu.  
The Calculate menu is located at the top of the Assembly pane.
4. Click Volume
5. Drag the label and click to set the position.

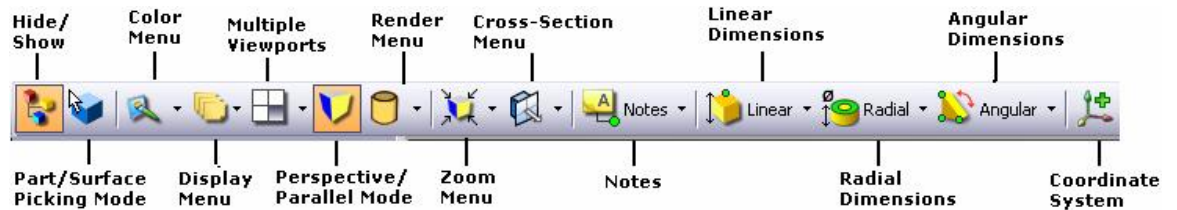


**Note:** You can alternate among a Part Volume, Part Surface, or Part Center of Mass label by **right-clicking** the label and selecting the new measurement from the menu.

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##### **To measure the Area of a single Surface:**

1. Select the **Assembly** pane.
2. Select the *Surface Selection Mode* Icon. 
3. Select the surface on the part that you would like to measure.
4. Open the **Calculate** menu.
  - The Calculate menu is located at the top of the Assembly pane.
5. Click *Surface*
6. Drag the label and click to set the position.




**Note:** You can select multiple surfaces using the <Ctrl> key on your keyboard

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### Part Extent / Scene Extent Box


The *Part Extent Box* command draws a bounding box around a single part and measures the part along the X-, Y-, and Z-axes.

#### **To measure the extents of a single part:**

1. Select the **Assembly** pane.
2. Select the part, assembly, or sub-assembly that you would like to measure.
3. Click the Extent Box  button.
  - This button is located at the top of the Assembly pane.




#### **To measure the extents of a Model:**

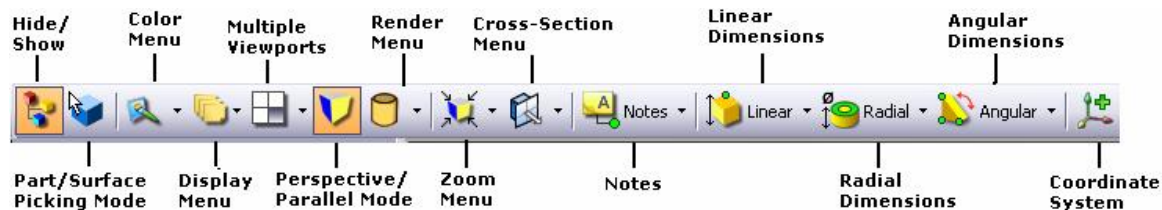
1. Select the **Assembly** pane.
2. Click **Scene**.  
If you would rather measure the extents of a part or subassembly, then click that object instead.
3. Click the Extent Box  button.
  - This button is located at the top of the Assembly pane.

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### Radial Dimensions

#### **To measure an arc diameter or Radius:**

1. Open the Radial Dimension menu  on the Viewer toolbar.  
*Click* the arrow to the right of the **Arc** button. You should see a drop-down menu listing the available commands for *Diameter* and *Radius*.
2. *Click* "**Arc Diameter or Arc Radius**".
3. Position your pointer over the arc you want to measure in the model.  
SpinFire Professional indicates when your pointer is over an arc by highlighting the arc with a circle.
4. *Click* on the arc.
5. Drag the label into position and *click* once.
  - Note: You can also Right-Click on the new Dimension and display either *Radius*, *Diameter*, or *Length*



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### Distance between (2) Points

The **Vertex to Vertex** Distance command measures the distance between two vertex points you select.

#### ***To measure the distance between two vertices:***

1. Open the **Linear** menu on the Viewer toolbar.
2. **Click** "Vertex to Vertex" and one of the (4) options".
  - Red dots designate available vertices on the model. A cross-hatch indicates when the pointer is properly positioned over a vertex point.
3. **Click** two vertices on the model.
4. Drag the label into position and **click** the mouse.

### Arc Center to Arc Center

The "**Arc Center to Arc Center Distance**" command measures the distance between the centers of two existing arcs on the model

#### ***To measure the distance between two arc centers:***

1. Open the **Linear** menu on the Viewer toolbar.
  - Click the drop-down arrow to the right of the *Linear* button. You should see a drop-down menu listing the available commands.
2. Point to **Arc Center to**.
3. Click **Arc Center**.
4. Click the first arc you want to measure in the model.
  - SpinFire Professional indicates when your pointer is over an arc by highlighting the arc with a circle.
5. Click the second arc.
6. Drag the label into position and click once.

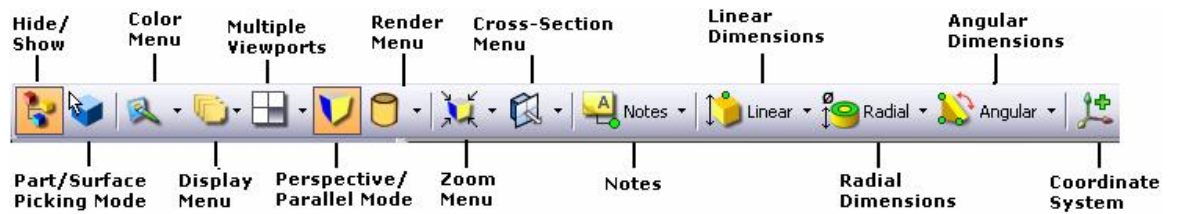
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### Sum of Connected Edges

The **Sum of Connected Edges** command measures the sum of any number of connected edges selected by the user.

#### ***To measure the sum of connected edges:***

1. Open the **Linear** menu on the 3D Document toolbar.



- Click the drop-down arrow to the right of the Linear button. You should see a drop-down menu listing the available commands.
2. Click **Sum of Connected Edges**.
  3. Click each connected edge that you would like to measure.
    - SpinFire Professional indicates when your pointer is over an edge by highlighting the edge with a line.
  4. Click one of the already selected edges to complete the selection process and display the measurement label.
  5. Drag the label into position and click once.
  - 6.

**New For 8.0** **Measure a Boundary Center by Edges**

The **Boundary Center by Edges** measures the center point of a boundary defined by connected edges.

***To measure a Boundary Center by edges:***

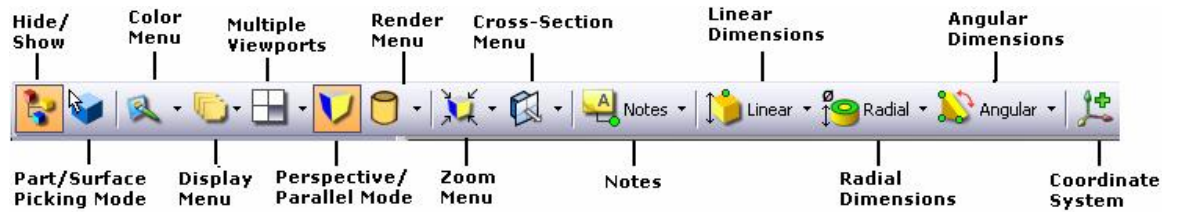
1. Open the **Linear** menu on the 3D Document toolbar.
2. Click **Boundary Center by Edges**.
3. Click an edge or arc on the assembly.
4. If necessary, continue clicking connected edges or arcs on the assembly.
  - When SpinFire Professional can determine the edges of a boundary, the center point will be measured.
5. Drag the label into position.

**Measure a Minimum Distance Between Edges**

The **Minimum Distance Between Edges** command measures the minimum distance between two edges.

***To measure the minimum distance between two edges:***

1. Open the **Linear** menu on the 3D Document toolbar.
2. Click **Edge to Edge**.
3. Click an edge on the assembly.
4. Click another edge on the assembly.
5. Drag the label into position and click once.



## V. Note Functions

### Add Notes

SpinFire Professional offers five types of notes:

- **3D Vertex Note:** Attached to a vertex point on the model.
- **3D Edge Note:** Attached to a straight line on the model.
- **3D Note:** Attached to any point in the 3D space.
- **2D Note:** Not attached to any point in the 3D space. Label remains stationary.
- **3D Surface Note:** Attached to any point of a surface.

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### Add Note Hyperlinks to Views and Workspaces

A markup label can be made into a link so that a user can access a file, Web page, or *User Defined View* by clicking the label. A markup label that's a link will be underlined.

Examples of valid link paths include:

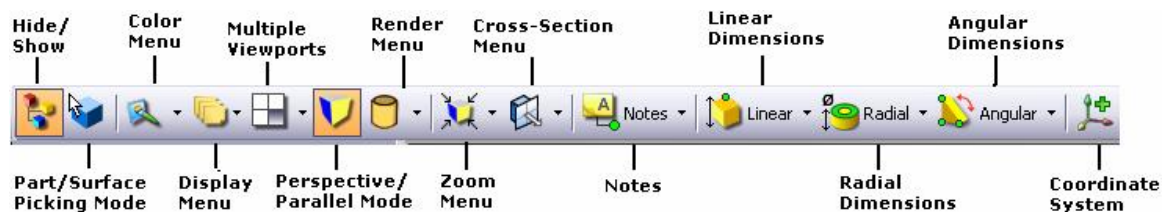
- <http://www.actify.com>
- C:\MyDocs\SomeDocument.doc
- #view2 – Creates a link from the Note to the *User Defined View* labeled “view2”
- [2D Markup] - Creates a link from the Note to the Workspace labeled “2D Markup”
- [3D Model]#view2 - Creates a link from the Note to the Workspace labeled “2D Markup” and positions to the User Defined View labeled “view2”

To link to a user view, type the pound sign # and the name of the saved view.

To link to another document within the .3D file, surround the name of the document with square brackets [ ]. For example, a link [Wheel] would open the Wheel document.

If you would like to link to a **user view** within a 3D document add the pound sign # and the view name. For example, the link [Wheel]#view2 would open the Wheel document and display the view named view2.

SpinFire Professional will append the http:// protocol header to any relative link. Therefore, links to files must use an absolute path.



### To add a link to a markup label:

1. **Right-click** the label.  
You can do this on either the label itself or the name of the label in the Markups tab. This opens the Markup context menu.
2. **Click** Properties on the menu.  
This opens the Markup Properties dialog box
3. **Click** in the Hyperlink field.
4. Type the path of the Web page or file you want to link to. Or **click** the [...] button to browse to a local file.
5. **Click** OK.  
The text of the label should be underlined.

## VI. User Defined / Explode Views

### User Defined View

You can save any *Camera Angle* of your model and choose from previously saved "user views" from the View tab in Viewer.

A user view may consist of just the position of the "camera" or it might also include part rotations and translations that are specifically applied to that user view.

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A "user view" may also store the State of the Parts selected. That is, it will store the Color, Render Mode, as well as Visibility of the Parts.

### To save a view:

1. Position your view of the model to the one you want to save.
2. **Click** the Views tab.

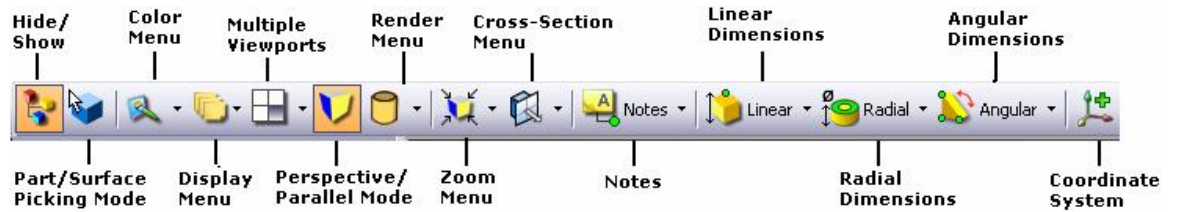


3. **Click** the New... button.  
This opens the New User View dialog box.
4. Type a name for your view and **click** OK.  
Your view will appear under "User Views" along with your username and the time it was saved.

### Create Exploded View Option #1

You can create an "Explode View" animation by doing the following.

1. First create a User Defined View and call it "**Assembled View**"
2. **Right-click** the Part or Subassembly in the Assembly Tree or on the Graphics Window



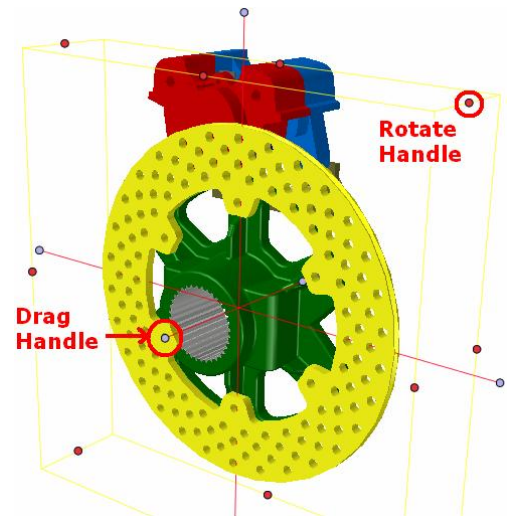
3. **Point to Transform**
4. **Select Translate**
5. From the "**Relative to:**" drop-down menu, select the *Coordinate Systems* you want to translate with respect to
  - The default is the current active *Coordinate Systems*.
6. Type the movement value in the proper fields
  - The units can be positive or negative
7. **Select Preview** to see what the translation will look like.
  - The object will briefly be translated to the new position and then will return to its original location.
8. **Select Apply->OK** to save the translation
9. **Create a New User View** named "**Explode View**"

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### Create Exploded View Option #2

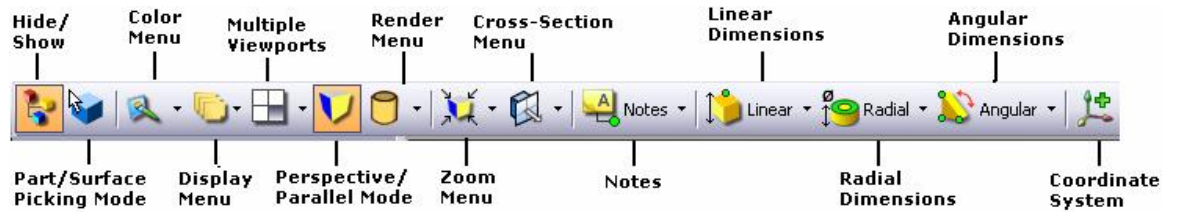
You can create an "Explode View" animation by doing the following.

1. First create a User Defined View and call it "**Assembled View**"
2. **Right-click** the Part or Subassembly in the Assembly Tree or on the Graphics Window
3. **Point to Transform->Handles**
  - This opens the Transformation Control dialog box and displays a box with handles around the object.
4. From the **Relative to:** drop-down menu, select the coordinate system you want to translate with respect to.
5. Drag one of the purple handles to translate the object.
  - A label will display the distance that the object is being translated.
6. Click **Apply** to save the translation.
7. Click **OK** to close the dialog box.
8. **Create a New User View** named "**Explode View**"



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Now, select the Views tab and select the "**Assembled View**" view just created. You will see the Part move as defined. Now, select the "**Explode View**" view to explode the parts.



To add additional parts to the **Explode View** follow the instructions above. However, instead of adding an additional View, just expand the “**Explode View**”, Right-Click on *Geometry*, and select *Update Geometry*.

**New For 8.0** **Create a View that Hides and/or Changes Part Color**

You can create an view that shows Parts Hiding, reappearing, and/or changing Color by doing the following.

1. First create a User Defined View and call it "**Start**"
2. **Right-click** the Part or Subassembly in the Assembly Tree or on the Graphics Window
3. **Translate** the part as described previously
4. **Right-Click** on the Part(s) or assembly
5. **Hide** the Part(s) or change their Color
6. **Create** a New *User View*

The view created will now show the part(s) change their visibility based on your selections.

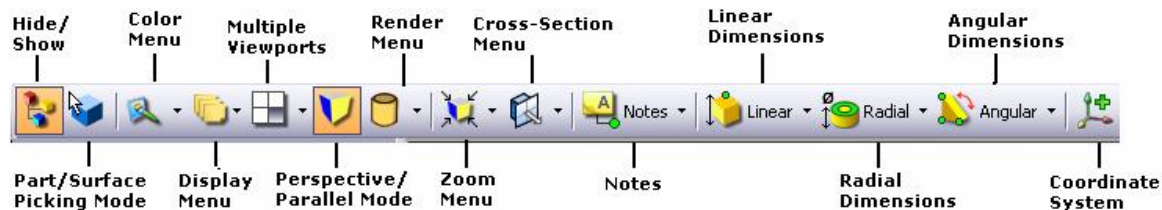
## VII. Section Cuts

The capability to display a cross-section of a part or assembly is one of the most important and useful features of SpinFire Professional. Mechanical designers look at cross-sections to have a better understanding of the structure of a part and of hidden components in an assembly.

### *Planes*

The Cross-Section Plane menu in the *Viewer toolbar* lists the available cross-section planes. You can recognize a cross-section plane by the eight handles that surround the rectangular plane.

SpinFire Professional offers standard planes bound by the X-, Y-, and Z-axes. Each of these planes is bound by two axes and can be translated, or shifted, along the third axis. For example, if you select "XY Section Plane," the resulting rectangular plane is defined by the X- and Y-axes; it can be rotated around the X- and Y-axes and translated along the Z-axis.



### New For 8.0 Enable/Disable the Cut of a Cross-Section

By default a cross-section plane cuts through the geometry in a 3D scene so that the material on one side of the plane is visible and on the other side is invisible. This behavior may be enabled or disabled through the Cut Material option.

A cross-section plane must be enabled for the cross-section profile to be available.

#### To enable/disable the cut of a cross-section:

1. Open the Cross-Section menu on the 3D Document Toolbar.
  2. Click **Cut Material**.
- OR
1. **Right-Click** on the Part or Subassembly
  2. **Select Section**
  3. **Select** Cut Material

### Change the Color of a Cross-Section Profile


New For 8.0 You may set the color of the Cross-Section Profile that appears on a Cross-Section plane.

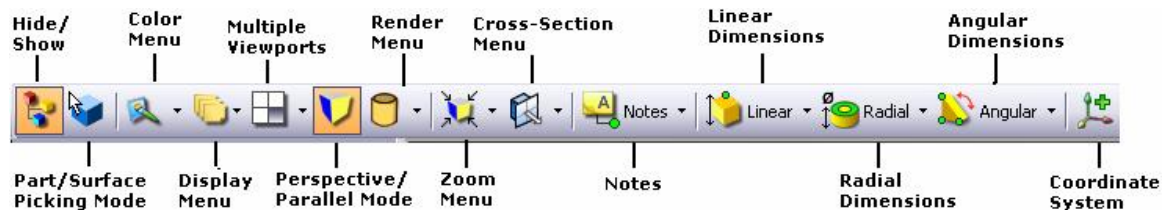
1. Open the Cross-Section menu on the 3D Document Toolbar.
2. Click **Profile Color...**
  - This opens the Color dialog box.
3. Select a color.

## VIII. Coordinate Systems

### Create a New Coordinate System

A 3D scene may incorporate multiple Axes by creating multiple *Coordinate Systems*. You will need to set the position and orientation of any new *Coordinate Systems* you add to the scene. The default settings of a new *Coordinate System* are equal to those of the active *Coordinate Systems*.

1. **Select** the  icon on the Toolbar  
This opens the *Coordinate Systems* context menu.
2. In the Name field, type a name for the *Coordinate System*.
  - This dialog box contains a Position tab and Orientation tab. The Position tab should be selected.
3. Choose one of two methods to set the origin point:



### Picking a point on the model:

- a. From the drop-down menu, choose a method for setting the origin.  
The methods for determining the origin point are:
  - **Pick a Vertex:** Select an available vertex point on the model.
  - **Pick a Circle:** Select a circle on the model. The origin will be set to the circle's center point.
 If the method you would like to select is already visible in the drop-down menu, you may simply *click* the Pick button.  
The Pick button should appear pressed in before you pick an element in the scene.
- b. *Click* the desired location in the 3D scene.

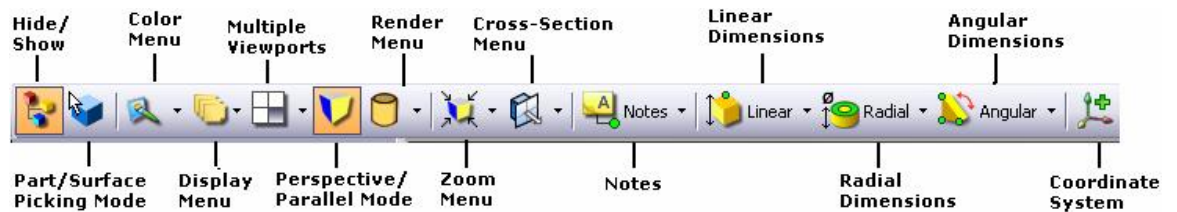
—OR—

### Offsetting the origin with respect to another *Coordinate Systems*:

- a. *Select Coordinate Systems* from the Relative To drop-down menu.
  - The origin point of the new *Coordinate System* will be offset relative to the origin point of the *Coordinate System* in the Relative To field.
- b. Enter the number of units you want the origin point offset in the Position X, Position Y, and Position Z fields.
- c. *Click Apply*.
4. *Select* the Orientation tab.
  - This tab lets you set an axis of the *Coordinate System* equal to a vector. This can be done either by selecting a vector from the 3D scene or by entering vector values in the dialog box.
5. *Select* the option button for the axis you want to orient.
  - The options buttons are the round buttons to the left of the X-Axis, Y-Axis, and Z-Axis options. The selected axis will be oriented.
6. Choose one of two methods to set the orientation of the axis:

#### **Selecting a vector in the 3D scene:**

- a. From the drop-down menu, choose a method for setting the axis.  
The methods for determining the orientation of an axis are:
  - **Pick 2 Vertices:** Select two available vertex points on the model. The axis direction will be toward the first point picked.
  - **Pick an Edge:** Select an edge on the model. When the pointer is properly positioned over an edge, the edge will be highlighted on the model.
  - **Pick another *Coordinate System* Axis:** *Click* the axis of another *Coordinate System* in the scene.
  - **Pick 3 Vertices (Normal Plane):** Pick three points to define a plane.



If the method you would like to select is already visible in the drop-down menu, you may simply **click** the Pick button.

The Pick button should appear pressed in before you pick an element in the scene.

b. **Click** the desired location in the 3D scene.

—OR—

**Entering vector values in the dialog box:**

a. From the Relative To drop-down menu, choose the *Coordinate System* you would like the vector to be relative to.

b. Enter values in the x, y, and z fields next to the selected axis.


c. **Click** Apply.

7. **Click** OK to save the settings of the *Coordinate System* and close the dialog box.

## IX. Library Component

Commands for working with local and remote files are available from the File context menu. Open the File context menu by **right-clicking** a file in the right pane of the Library. The commands listed on the Folder context menu will differ slightly depending on the type of file.

### Add a Local Folder to Your Library

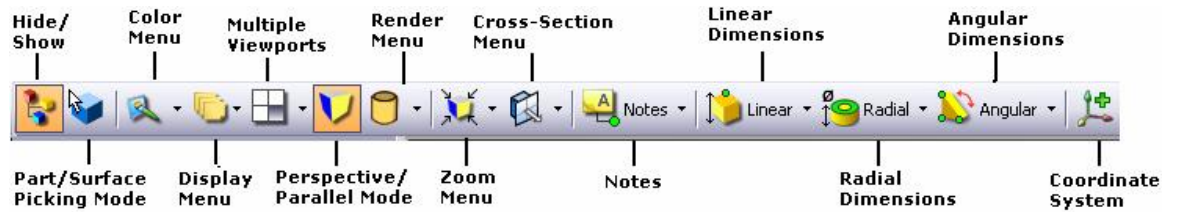
Local folders can include any folder on your hard drive or network. Local folders are identified by a folder icon.  This is a much easier method than performing the File->Open->Browse selection process.

### To add a local folder to your Library:

1. Select the Library component.
2. **Click** the "Add Local Folder" button in the Library toolbar.
  - This opens a dialog box listing your local and network folders.
3. Navigate through the directory tree and **click** the folder you want to add.
4. **Click** OK
5. This new folder should now be listed on the left under "Connections"
6. If there is a "+" sign next to the folder this means that there are subfolders.
7. Select "+" sign to expand the folder

### Add multiple files to your SpinFire Session:

1. Start SpinFire



2. Select the "Library" Component on the Left side of the SpinFire screen
3. If the "Folder" where your CAD files are located is not listed, see "Add Local Folder" above
4. Now, locate the CAD file in the listing on the right
5. **Double-click** to Import this file

**To Add a 2nd file do the following;**

1. Select the "Library" Component on the Left side of the SpinFire screen
2. Locate the 2nd file you want to add
3. **Right-click** on this file and select "Import"

**X. 2D Snapshot**

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**Take a 2D snapshot of a 3D model**

SpinFire Professional allows you to take a 2D snapshot of a 3D model and work with it in 2D mode. The snapshot will be a 2D image of the current view of the model. You may take a snapshot and immediately open it in as a new 2D document (using the Snapshot button ). Or you may copy the snapshot to your clipboard and paste it into an existing open 2D document (using the Copy Snapshot and Paste Snapshot buttons).

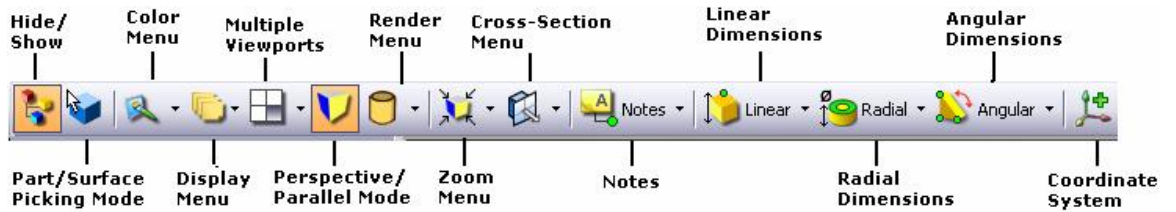
**To take a snapshot and create a new 2D document:**

1. Select the document where you would like to take a snapshot so that it is active. If multiple documents are open, you should **click** the tab of the document you want.
2. **Click** the Snapshot button on the General Toolbar and select *New Snapshot*.  
A new 2D document with the snapshot will be created.



**To take a snapshot and paste it into an existing 2D document:**

1. Select the document where you would like to take a snapshot so that it is active. If multiple documents are open, you should **click** the tab of the document you want.
2. **Click** the Snapshot button on the General Toolbar and select *Take Snapshot*.
  - This copies the snapshot to your clipboard.
3. **Click** the tab of the 2D document you where would like to paste the snapshot.
4. **Click** *Insert Snapshot* to insert the new image



## XI. Importing Multi-document .3D

New For 8.2

When importing a multi-document .3D file into an existing document, a dialog box will be displayed that enables the user to choose the operation to perform on each document in the file:

- **Add:** The document will be added to the WorkSpace. A new tab with that document will be created.
- **Import:** The document will be imported into the current active document: the scene will display the geometries of both the documents. This option is available only for 3D documents
- **None:** The document will be skipped.

