DesignSpark Mechanical FAQ

Q. How do I change my sketch grid spacing?

A. In the Grid group in the Units screen in DesignSpark Mechanical Options, enter your grid spacing preference.

Q. How do I make my model transparent, but not wire frame?

A. After selecting the solid, try the following:

- Select Color from the Design group, and then click the Opacity bar to change the opacity of the model.
- Use the Color or Style Override tools, located in the Style group on the Design tab to set opacity for selected objects. Style Override makes the object opaque or transparent, regardless of face or layer settings. These overrides are automatically deselected when you change the transparency of an object using the Color tool or Style Painter.

Q. Why are some objects transparent and others opaque?

A. By default, surfaces are transparent and solids are opaque.

Q. How can I improve graphics quality?

A. In the Graphics performance options group in the Popular screen in DesignSpark Mechanical Options, select a higher number Rendering quality. You can choose any number between 1 and 7, where 1 is the lowest rendering quality and 7 is the highest, but be aware that a higher setting may impact performance. Click the Recalculate Rendering button to force recalculation of all surface and line rendering.

Q. How can I improve performance?

A. In the Graphics performance options group in the Popular screen in DesignSpark Mechanical Options, select a lower number Rendering quality. You can choose any number between 1 and 7, where 1 is the lowest rendering quality and 7 is the highest, but be aware that a lower setting may impact graphics quality. Click the Recalculate Rendering button to force recalculation of all surface and line rendering.

Q. How do I save models with external files?

A. Click Save to save the active design as a DesignSpark Mechanical document. A top level assembly file will be created, along with a separate file for each external component.

Q. How do I work in a collaborative environment, with multiple people working from different environments on the same model?

A. Try the following:

- After making any changes, click the Application Menu, then select Save As > Save As New Version. Different users can now track progress on the model based on the versions you apply.
- Agree that only one person work actively on the model at a time, which allows you and others to see each iteration model as it develops.

Q. Can I snap to a grid while using the Pull or Move tool?

A. Yes. In the Solids group in the Snap screen in DesignSpark Mechanical Options, you can change snap options for both incremental or objects settings.

Q. How and why should I use the Groups feature?

A. Use groups to define a set of selected objects or to setup driving dimensions.

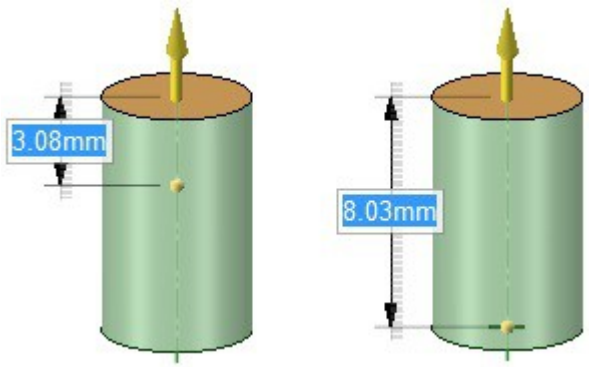
- Select a set of objects and click Create Group in the Groups panel to save the selection for future use.
- Create a ruler dimension in the Pull or Move tool and click Create Group to save that dimension as a driving dimension.

Q. How can I create a surface patch?

A. Use the Patch blend option within the Fill tool in the Edit group on the Design tab. You can select faces and the Fill tool will automatically create a patch if you also select at least one edge.

Q. How can I dimension my model?

A. First, determine if you are dimensioning the model to indicate size and proportion or, if you want to drive changes to the model with dimensions. Both scenarios are described below. Changes to a model are done with tools that use dimensions to define the size of a change. Usually dimensions appear on screen when using a tool to make an edit. For example, when using the Pull tool to change the height of a cylinder, a dimension appears that shows the size of the change:



To create a dimension that measures the height of the cylinder, click the Ruler icon in the Options panel or on the mini-toolbar. Click the bottom of the cylinder to create a temporary dimension that measures the height. Creating dimensions for reference is done with DesignSpark Mechanical Annotation tools. Use the Dimension tool on the RS/Allied tab to create dimensions that measure length, angle, and other characteristics.

Q. How can I render my model?

A. Rendering is not part of the DesignSpark Mechanical software at this time.

Q. How can I specify a shape for the new surface I create using the Patch Fill tool?

A. Patch Blend in the Fill tool uses faces, curves, and points as input when creating a surface patch. The created patch attempts to fit through selected faces, curves, or points.

Q. How do I constrain my sketch?

A. Sketches are not constrained in DesignSpark Mechanical because they are not intended to persist. Create a sketch to the correct dimensions, turn it into a surface, and then use the Pull tool to pull it into a solid. For best results, make edits to the resulting solid rather than editing the sketch.

Q. How do I create a fillet?

A. A fillet is a round in DesignSpark Mechanical. To create a round:

- Click the Pull tool in the Edit group on the Design tab.
- Select the edge or edges you want to round.
- Select the Round option in the Options panel or from the mini-toolbar.
- Click and drag the edge in the direction of the Pull arrow.

Q. How do I create a helical revolve?

A. To create a helical revolve:

- Select the Pull tool.
- Select the face or edge you want to revolve.
- Alt+select a line or an axis to revolve about.
- Click the Revolve helix option in the Options panel and enter dimensions for the helix.

Q. How do I create a pattern?

A. To create a pattern:

- Click Move in the Edit group of the Design tab.
- Select the Create patterns checkbox in the Options -Move panel.
- Select a protrusion, depression, sketch, points, axes, planes, origins, or 3D curves to be the first member of the pattern.
- Drag a move handle to copy the first pattern member to the location of the last member of the linear pattern.
- Type a new value for the pattern count.
- Press Tab to change the distance or spacing.

Q. How do I create a rotational pattern?

A. To create a rotational pattern

- Select the geometry you want to pattern.
- Click the Move tool.
- Select the Create Pattern checkbox in the Options panel.
- Anchor your move tool to the center point you want to pattern around.
- Rotate the Move tool to create your pattern.

Q. How do I dimension my model?

A. To create dimensions for annotating a part, use the Dimension tool in RS/Allied tab.

- All dimensions are placed on an annotation plane in design mode, which you can toggle on or off for display purposes.
- You can use the Ruler option in the Pull or Move tool to create temporary dimensions while editing a part. You can save these dimensions by creating a group in the Groups panel. These dimensions can then be used to drive changes with the Pull or Move tool.

Q. How do I get back to my sketch?

A. DesignSpark Mechanical does not save sketches for future edit. Once you create a solid from a sketch, DesignSpark Mechanical recommends that you use the Pull and Move tools to edit the solid instead of editing the sketch.

Q. How do I make assembly models?

A. You make assembly models by placing each solid in a component. Simply right click on a solid in the Structure tree and select Move to New Component, and then name the component. Components can take on assembly conditions, whereas solids by themselves cannot have assembly conditions. With click and drag, components can be rearranged to reflect any desired assembly structure.

Q. How do I package multiple components into one assembly to send to a client?

A. In order to send a client an assembly model properly, you need to make sure that all your components have been internalized. Click on a component in the Structure tree with the right mouse button and select Use Internal Copy to make it internal. Finally, save your assembly and send the resulting DesignSpark Mechanical document to the client.

Q. How do I use the Wrap around target option in the Project tool?

A. The Wrap around target tool allows you to take planar curves or text and wrap them onto planes, cylinders, and cones. You may choose a surface target or establish a direction for the wrap. Curves can be wrapped onto geometry with per-existing imprinted wrapped curves. Surfaces may be used for wrapping, however only the surface edges are used to wrap around the target.

To wrap a curve, text, or a surface around a target object:

- Click the Project tool
- Check ON the Wrap around target option
- Click the curve, text or surface you want to wrap
- Click the Select Direction tool guide and choose a direction, you will see purple lines on the target as a preview
- Click the Complete tool guide or press the Enter key

Q. What does Style Painter do?

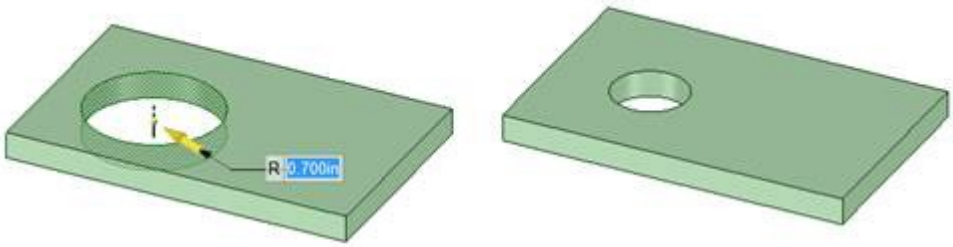
A. Style Painter paints display properties from one object to another. The tool applies color and transparency intelligently from one object type to a different object type, for example, you can copy body colors and textures between geometry or font styles between notes and annotations.

Q. What is the difference between the Mirror tool in Sketch mode versus 3D mode?

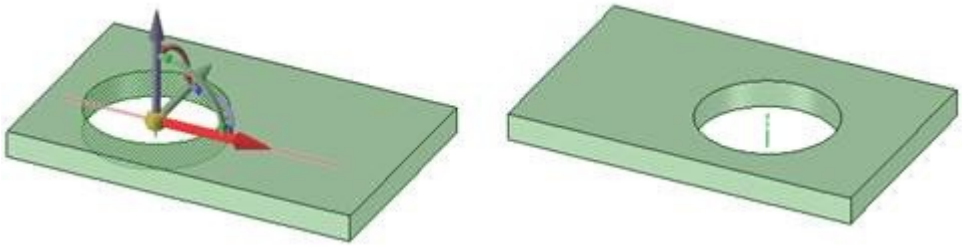
A. You can use the Mirror tool to mirror bodies, faces, and sketch curves. The Mirror tool can be used in Sketch mode or in 3D mode. In Sketch mode, you can also create a mirror line: draw a line and select Set as Mirror Line from the right mouse button context menu. If there is a mirror line in your current sketch plane, all new sketches will mirror across the line. In addition, if you make a change to a mirrored curve in the sketch plane it will modify the curve and its mirror.

Q. What´s the difference between Pull and Move?

A. In the simplest sense, Pull creates or extrudes, and Move translates or rotates. For example, if you pull a hole, the size of the hole changes:



But if you move the hole, the location of the hole changes:



Only in the case of Pulling or Moving a planar face is the behavior similar.

Q. When I try to pull multiple faces, why do I get surfaces instead?

A. This may occur if you continue to hold the Ctrl key. Try the following:

- Hold the Ctrl key to make a copy of a face as you drag.
- Hold the Ctrl key to select multiple faces, and then release the Ctrl key before using the Pull or Move tools.

Q. Where do I perform Boolean operations?

A. Use the Combine tool in the Intersect group on the Design tab to perform Boolean operations.

Q. Why are solids grayed out in the design window?

A. The solids may not be included in an active component. To activate the document level component, click the right mouse button on the top level component and choose Activate Component.

Q. Why can´t I combine a solid with a surface?

A. The surface must extend through the solid. Use the Pull tool to extend the surface until it fully intersects the solid.

Q. Why does a face change in size when I pull it?

A. Most likely you are pulling on a face without also selecting its neighboring edges. Try selecting the edges along with a face and then pull. The face stays the same size and extrudes that geometry. By default DesignSpark Mechanical uses the adjacent faces to influence how to pull a face.

Q. Why does my solid turn into a surface when I hit the Delete key?

A. The Delete key removes geometry from a model. If you delete a face, the solid turns into a surface with a hole where the face used to be. In this scenario, DesignSpark Mechanical recommends that you use the Undo tool to return your design to the solid state. If this is not possible, the Missing face tool in the Fix group on the Repair tab can help you search for holes in the model and will attempt to patch over them.

Q. Why does one solid in an assembly automatically move every instance of that component throughout the assembly?

A. If you have multiple placements of the same part (component containing a solid) in an assembly, also referred to as an instance, they will all share the same geometry. If you select a solid in one instance and move it, it is considered changing the geometry and the solid will move in each instance. You should select a single component in the structure tree and move it only if you want to change the location of that instance.

Q. Why won´t solids combine together?

A. The operation may fail because of errors in the body geometry. You must remove the bad geometry before attempting to combine solids. Select the solids, and then select Check Geometry in the Inspect group of the Measure tab to identify any bad geometry.

Q. Can I filter my selection?

A. Yes. Click the white cursor arrow in the bottom right corner of the design window on the status bar to display the selection filter, which is a drop down menu that shows all selectable entity types. Turn off Smart selection, then toggle which entity types you would like to remain in your selection. Changing tools turns the filter back to Smart. DesignSpark Mechanical retains these settings the next time you enable the selection filter.

Q. Is there an easy way to select multiple faces on a model?

A. Yes. You can select one or more objects quickly and easily by using the Select tool. Click the first face of your design, hold down the Ctrl key, and then click each individual selection you want to add to or remove from your original selection.

Q. Can shortcuts be customized?

A. No. Keyboard shortcuts cannot be customized.

Q. How do I change the Zoom/Spin/Pan controls from the default settings?

A. In the Navigation screen in DesignSpark Mechanical Options, you can set your keyboard and mouse controls. Select a control, choose an option from the control setting´s dropdown, and click OK.

Q. How do I rotate views without knowing which way is the top view, for example?

A. While in 3D mode, you can use the Snap View tool, located in the Orient group on the Design tab. You can click a face to view it head-on, then click, drag, and release the mouse towards the top, bottom, or sides of the design window to ‘throw´ the face to that side. To display a straight-on view of a planar face, select the face and click Plan view in the Orient group on the Design tab.

Q. How should I use the Properties panel?

A. The Properties panel shows information about a selected object. Use this panel to view and edit characteristics of an object. In addition to components, surfaces, and solids, you can use the Properties panel to modify property values for sketched objects, inserted images, patterns, and sheet metal.

Q. My DesignSpark Mechanical Options menu doesn´t display. How do I fix this?

A. Try the following:

- If you have multiple displays, check to see if the menu displays on another monitor.
- Try using a shortcut to move windows on to another display: Hold down the ALT and spacebar keys. Select Move, then press the arrow keys on the keyboard or move the mouse to bring the panel back into your preferred display.

Q. My DesignSpark Mechanical panels and tabs don´t display where I want them. How can I adjust them? How do I customize my window layout?

A. In the Appearance screen in DesignSpark Mechanical Options, click the Reset Docking Layout button to reset the layout. To customize the layout of your panels, click and drag a panel to a new location. You can also click and drag, and then pin, a panel to a DesignSpark Mechanical window edge.

Q. DesignSpark Mechanical looks like a Windows application; does it have similar cut, copy and paste functionality?

A. Yes. You can work with these functions from the Clipboard group on the ribbon tab, or, use shortcut keys such as Ctrl+C to copy, Ctrl+V to paste or Ctrl+X to cut.

Q. What´s the difference between a solid at the top level of the design and a solid within a component?

A. The major difference is that you cannot make assembly conditions between solids at the top level of the design. You can only do this with solids within components. Having components within your design:

- Confirms that you are making an assembly. You can move components around in the design window and also use components to make sub-assemblies of a larger model.
- Allows you to create assembly conditions. You cannot use assembly conditions between solids which are not in components
- Prevents solids from merging unintentionally in the Pull tool.

Q. Where is the feature tree?

A. DesignSpark Mechanical is a direct modeling software package, and as such, uses a Structure tree to list all of the objects in the design window. The Structure panel contains the Structure tree, which shows you each of the objects in your design. Objects are displayed in the order in which they were created, or, in the order in which they were imported.

Q. Where is the world origin located? Can I change its location?

A. World Origin shows the X, Y, and Z zero location in designs and drawing sheets. By enabling this feature, you can display the axes that set the default orientation of the design in the Design window. In DesignSpark Mechanical, you can turn on the world origin by selecting the World Origin checkbox in the Show group on the Display tab. The position of the world origin cannot be changed, but geometry can be moved up to, or dimensioned from the origin.