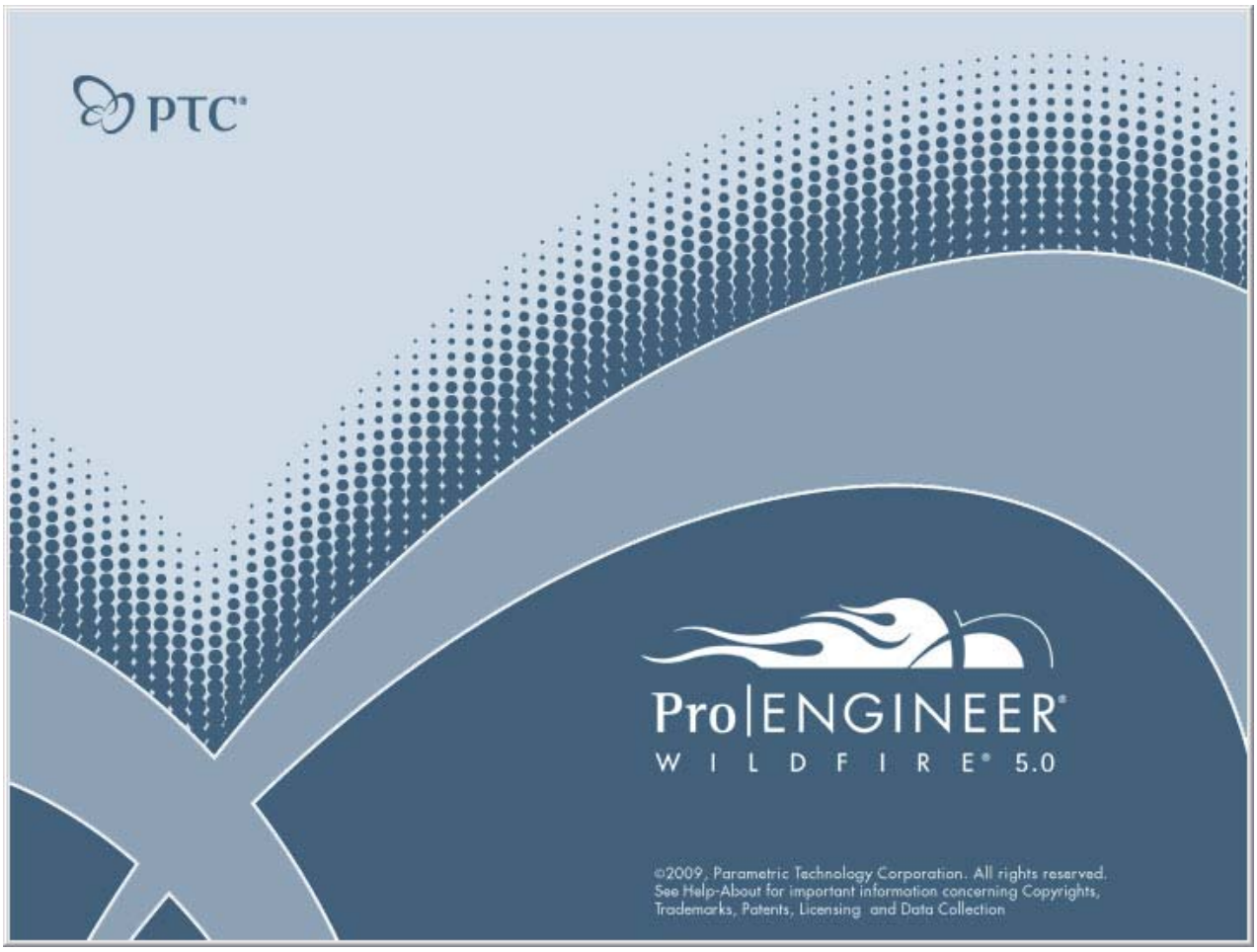
PTC/USER 2009
Pro/ENGINEER Wildfire 5.0
Hands On Workshop



WORKSHOP USE ONLY

Please do not write notes in this book or remove from the workshop. It will be used by all participants.

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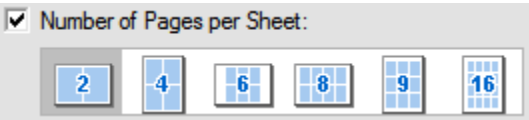
CAD Files

The CAD files required for this tutorial can be download from this location:
http://download.ptc.com/products/proe/wildfire5/tutorial/wf5_how_student_v2.zip

Document Format & Printing

This document is setup to be printed with (2) pages per sheet to minimize the total number of printed pages. If available, please setup your printer for multi-page printing.



Conventions

Information is provided at the start of many tasks.

Tips are provided along the way, with time-saving or alternate techniques.

Notes are provided with additional useful details, which may not be required to complete the tutorial.

- Menu commands are shown in **Bold**
- The comma character **,** is as a separator between commands
- Icons are shown in line with command text
- Keyboard keys are shown in Bold **CAPS**
- The left, middle and right mouse buttons are referred to as **LMB**, **MMB**, **RMB**

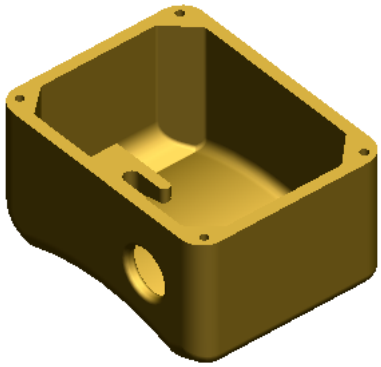
Interactive Modeling

This tutorial will cover some of the new sketcher, feature and edit functions in Pro/ENGINEER along with placement of User Defined Features (UDF) and the Model Properties dialog box.

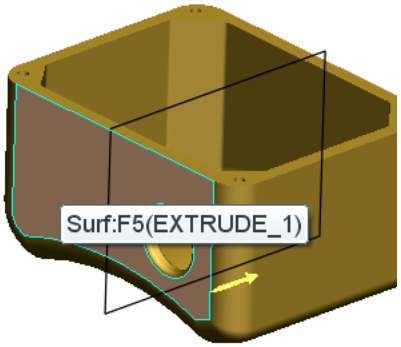
New Sketcher Functions

1. **File, Open**, **INTERACTIVE-MODELING** folder, select **housing.prt**, **Open**

Actual exercise model color may be different to improve contrast in sketcher

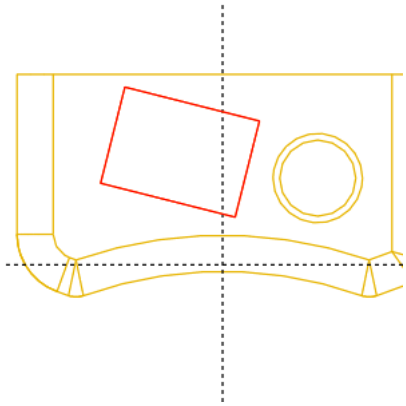


1. Select **Sketch**, select the surface shown below (with the hole) as the sketch plane, **Sketch**, toggle on **No Hidden**

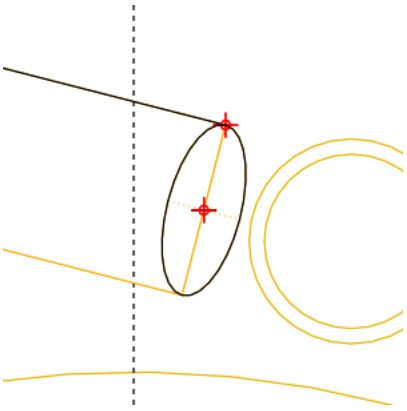


New sketching tools can directly create a slanted rectangle and a slanted ellipse, providing flexibility and speed in feature creation.

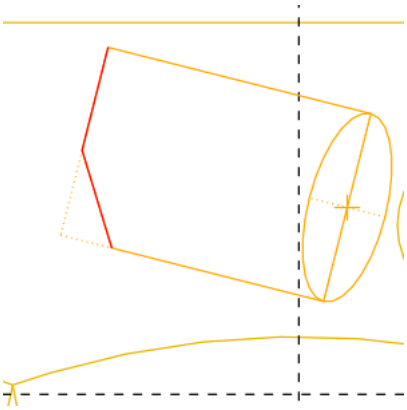
2. Select the fly-out arrow next to and select **Slant Rectangle**, sketch a slanted rectangle in the sketch plane, do NOT snap the rectangle to any references



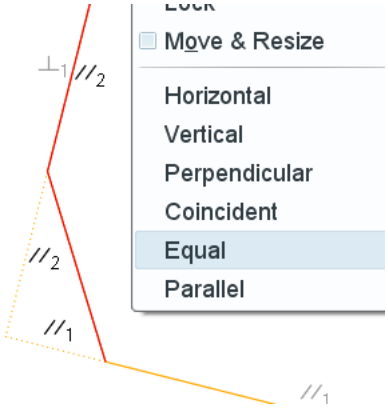
3. Pick the fly-out arrow next to and select **Center and Axis Ellipse**, start at mid-point of the width, end at the corner as shown, **LMB** to finish the ellipse



4. Select **Chamfer**, select the adjacent entities for length and width as shown below, **MMB** to finish

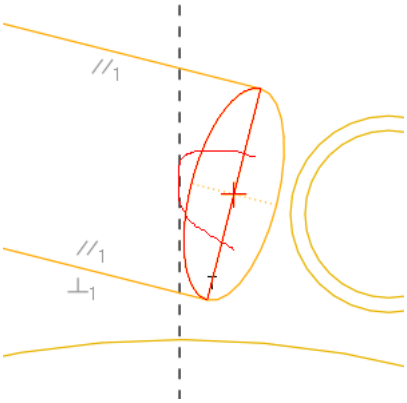


5. Hold **Ctrl** and select the chamfer and the adjacent entity shown, **RMB**, review all object-action constraints, **Equal**



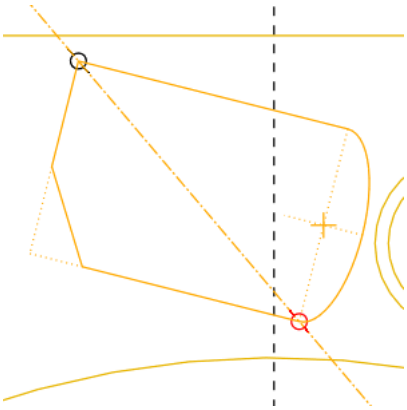
Sketcher constraints and workflows are more flexible. There are shortcut menus, object-action workflow, and a consolidated user interface.

6. Select **Delete Segment**, delete unnecessary sketch entities as shown

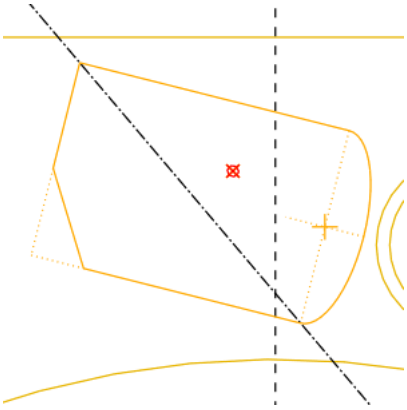


In the follow tasks, **make sure to use the fly-out to select Geometry Centerline or Point**. These are different than regular sketcher counterparts. Depending on how the sketch is used, they can produce datum features in the model.

7. Select the fly-out arrow next to and select **Geometry Centerline** , sketch a geometry centerline snapped to two corners as shown below

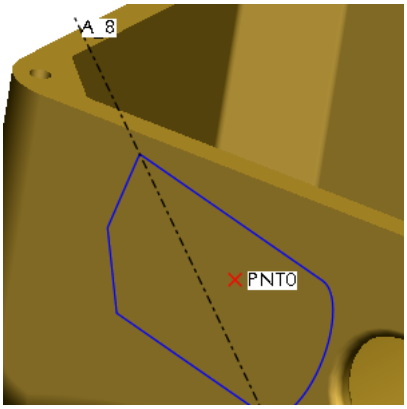


8. Select the fly-out arrow next to and select **Geometry Point** , sketch a point at any position inside the sketch, **MMB** to confirm

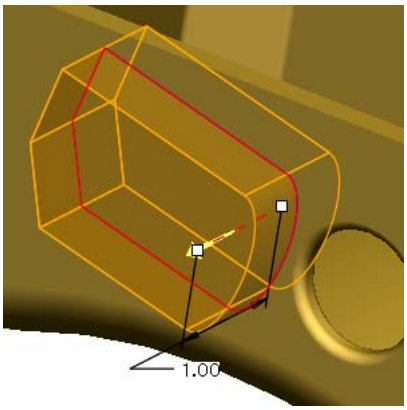


9. Select **Done** to finish sketch

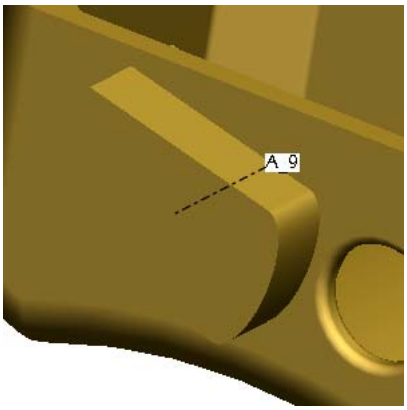
10. Toggle on **Shading** display, **Named View List**, **sketcher**, toggle on **Datum Axis** and **Datum Point**, review the datum axis and the datum point created in the Sketcher, toggle off **Datum Axis** and **Datum Point**



11. Pick **Extrude**, select the sketch just created, **RMB** on the depth control handle , **Symmetric**, change the depth to 1.00, **MMB** to finish



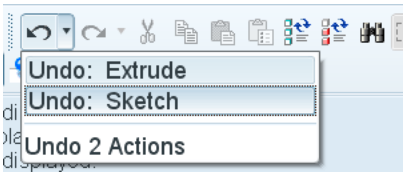
12. Toggle **Axis Display**, note that the Geometry Point resulted in a Datum Axis, toggle off the **Axis Display**



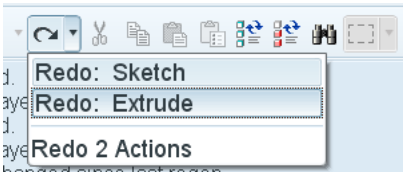
Undo/Redo

Some operations will clear the stack. If you do not see the exact list as shown below, experiment with a few operations and/or the choices available.

1. Select the arrow next to **Undo** and select **Undo: Sketch**



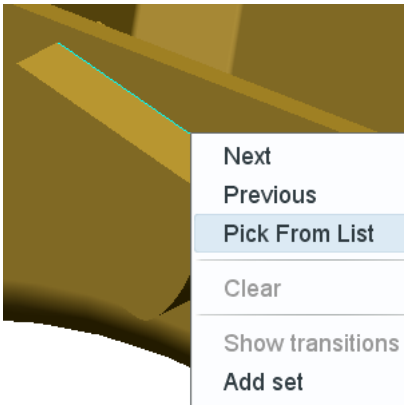
2. Select the arrow next **Redo** and select **Redo: Extrude**



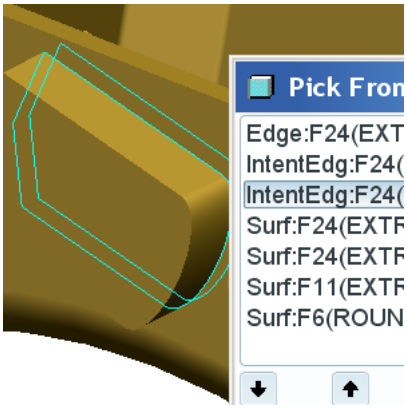
Dynamic Edit

You can use the Dynamic Edit command to edit features and immediately see the impact of dimensional changes on the model geometry. Use the 3D drag handles on the section to move the entire section. Note that constraints are enforced.

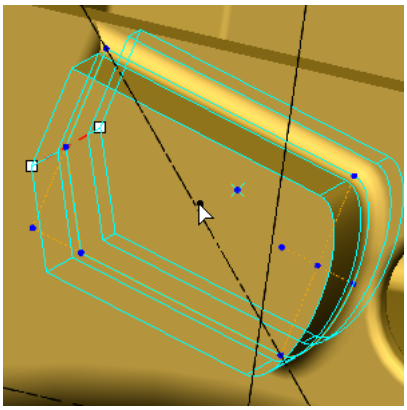
1. Select **Round** , **RMB** on an intersection edge between the extrude feature and the box wall, **Pick From List**



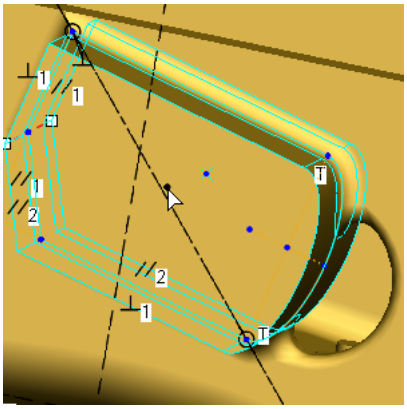
2. Select the **Intent Edges** from list for two loops from intersection of extrusion and the base model, **OK**



3. Change the round radii to 0.12, **MMB** to confirm
4. Select the extrude feature just created in the Model Tree or in the graphics window, **RMB Dynamic Edit**, select 3D drag handle where the cursor points as shown below



5. Move the entire feature towards the hole, review the dynamic changes

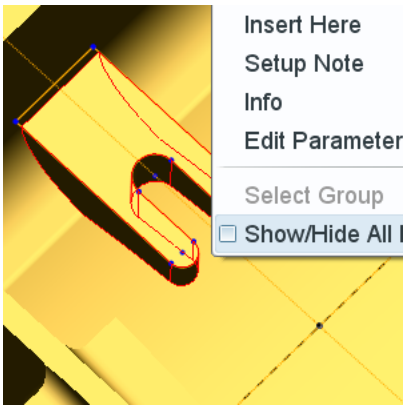


6. **LMB** on the graphic window to exit the dynamic edit

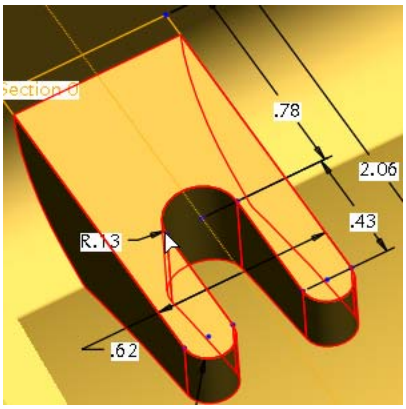
Failure and Diagnostics

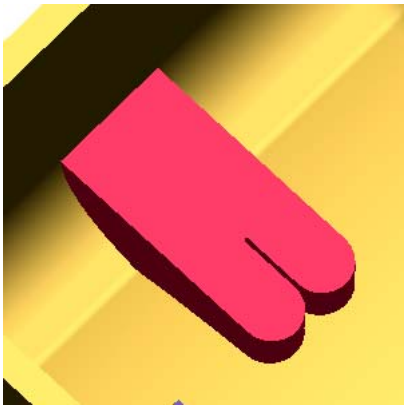
In Pro/ENGINEER Wildfire 5.0, you can deal with failures now or later, and models can now be saved with failed features. “Failed” geometry is shown when possible.

1. **Named View List** , **No_Resolve**, select the **BOSS_1** protrusion feature in the Model Tree, **RMB Dynamic Edit**, **RMB** in the graphics window, check **Show/Hidden All Dims**



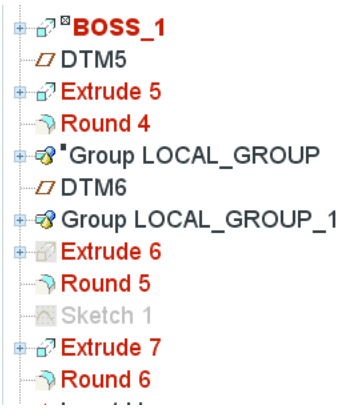
2. Drag the dim R.13 control handle where the cursor points, dynamically change the cut narrower until the feature fails and becomes red, **LMB** in the graphics window to exit





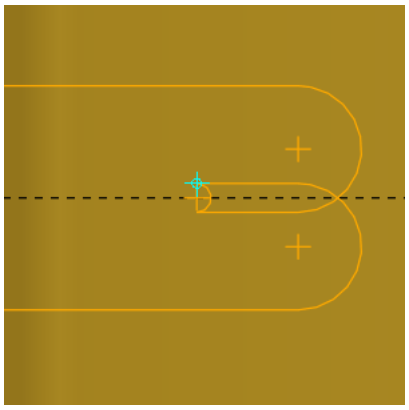
There is no resolve mode in Pro/ENGINEER Wildfire 5.0. You are given a warning and options to fix the failing feature(s).

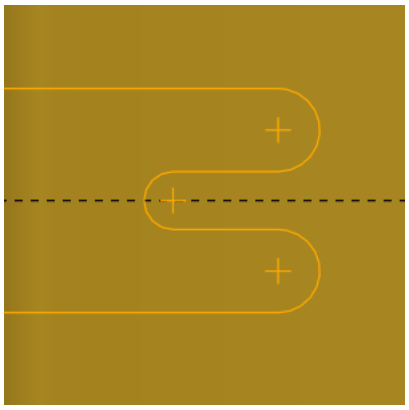
3. Click **OK** to accept the result and we will fix the fails later on, review the failed features highlighted in the Model Tree



4. Select **BOSS_1** feature in the Model Tree, **RMB Edit Definition**, click **Placement** in the dashboard, **Edit…**, drag and redefine the sketch until no intersected entities

Dynamic Edit can be used again for this step



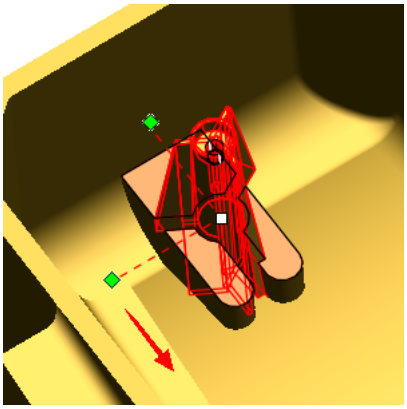


5. Select **Done** ✔, ✔ confirm, review the resolved features

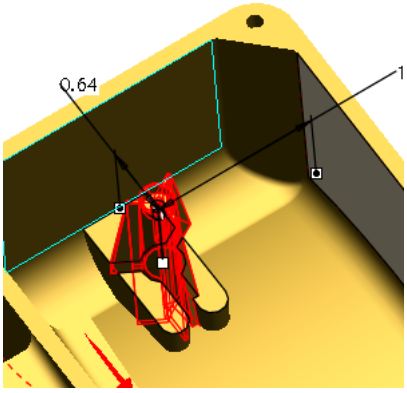
User Defined Feature (UDF)

You can place user-defined feature (UDFs) using the new on-surface coordinate system as a reference. You can preview the UDF geometry as it is being placed on the model, and also can see an immediate display of changes in variable dimensions and even specify additional rotations about the placement coordinate system

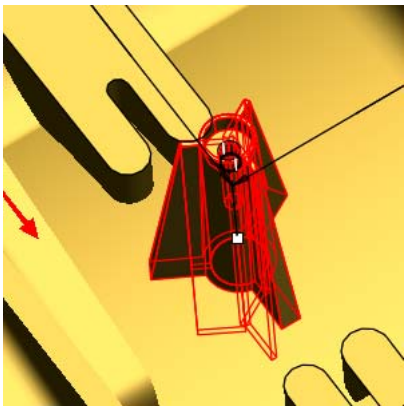
1. Toggle on **Csys Display**, **Insert**, **User-Defined Feature…,** browse to **INTERACTIVE-MODELING** folder and select **boss_udf.gph**, **Open**
2. Check **View source model** option, resize the **BOSS_UDF_GP** window, **OK**, pick the boss surface to specify the placement reference, toggle off **Annotation** display



3. Drag the two green placement handles to specify the location references



4. Drag the location handle and move, review the immediate udpates



5. Select **Variables** tab, change the rib_instance from 3 to 6, **Enter**, preview the changes of UDF boss in the graphics window

6. Select **Options**, **Adjustments** tabs to review options, select ✔, toggle off **Csys Display**

Model Properties

Model properties, such as materials, units, and accuracy, are located on a common Model Properties dialog box. This new dialog box also contains information on relations and parameters used in the model.

1. **File**, **Properties**, review all options of Model Properties
2. Select **Change** next to **Material**, **RMB** on the **steel.mtl** in the left column, **Assign**, **OK**, **Close** the Model Properties window

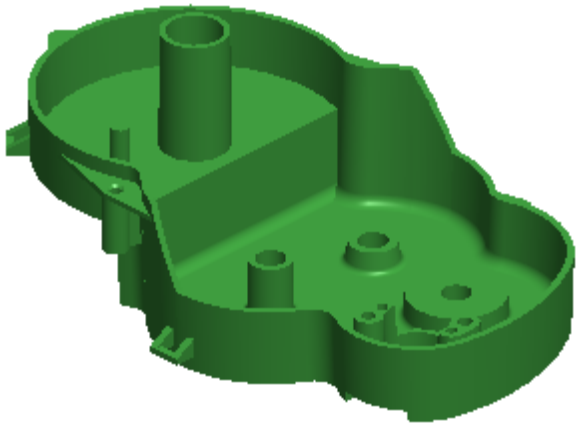
3. **Window**, **Close**
4. **File, Erase, Not Displayed**

Molded Part Design Efficiency

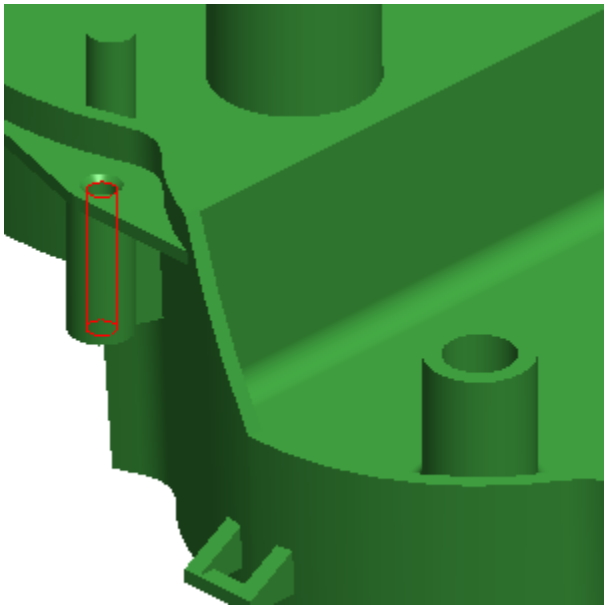
This tutorial will show some the new features and enhancements to assist in the design of cast and molded parts including geometry patterns, the new trajectory rib feature and enhancements to draft check analysis.

Pattern Enhancements

1. **File, Open**, **PART-MOLDED** folder, **gearbox.prt, Open**



2. Select **Hole 1** in the Model Tree**, Pattern**

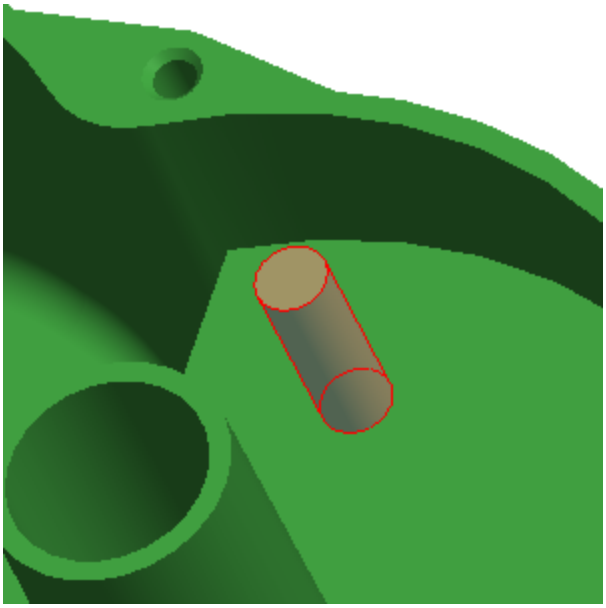


3. On dashboard, select **Point** from the drop-down list for type, then the **Use Points** option. In the Model Tree select **Datum Point 7012, MMB or**

4. In Model Tree, select **Chamfer 1**, **Pattern**, ensure that the type is **Reference**, **MMB** or

> In the next step, it may be helpful to change the smart filter to **Geometry**. Refer to the Quick Reference card for guidance on this type of selection.

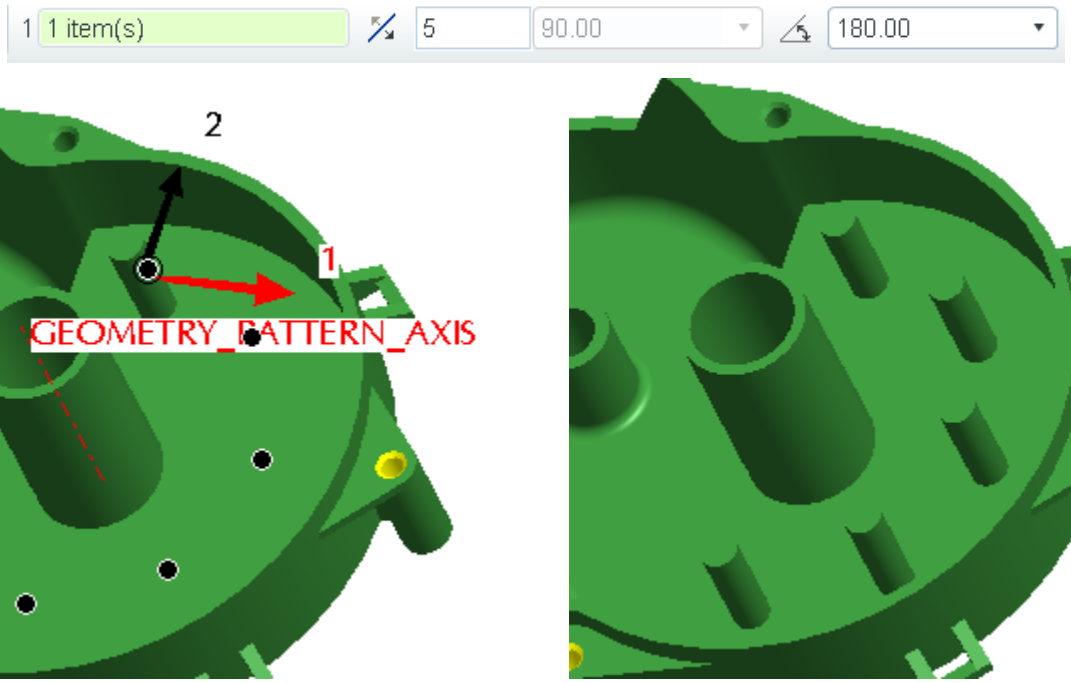
5. Select geometry using "surface and boundaries" technique. **Pick** top surface of the protrusion shown, then **Shift+Pick** the shell at base of the protrusion.



6. **Edit**, **Geometry Pattern**

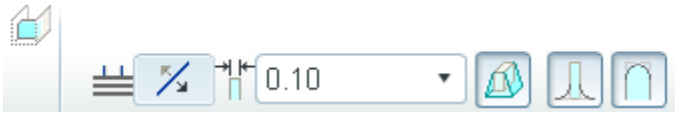
> Geometry patterns make regeneration faster as compared to patterning the entire feature definition.

7. Dashboard, **Axis,** select **GEOMETRY_PATTERN_AXIS** in Model Tree**,** enter **5** for number of pattern members, **Angular Extent**, enter **180**, **MMB** or

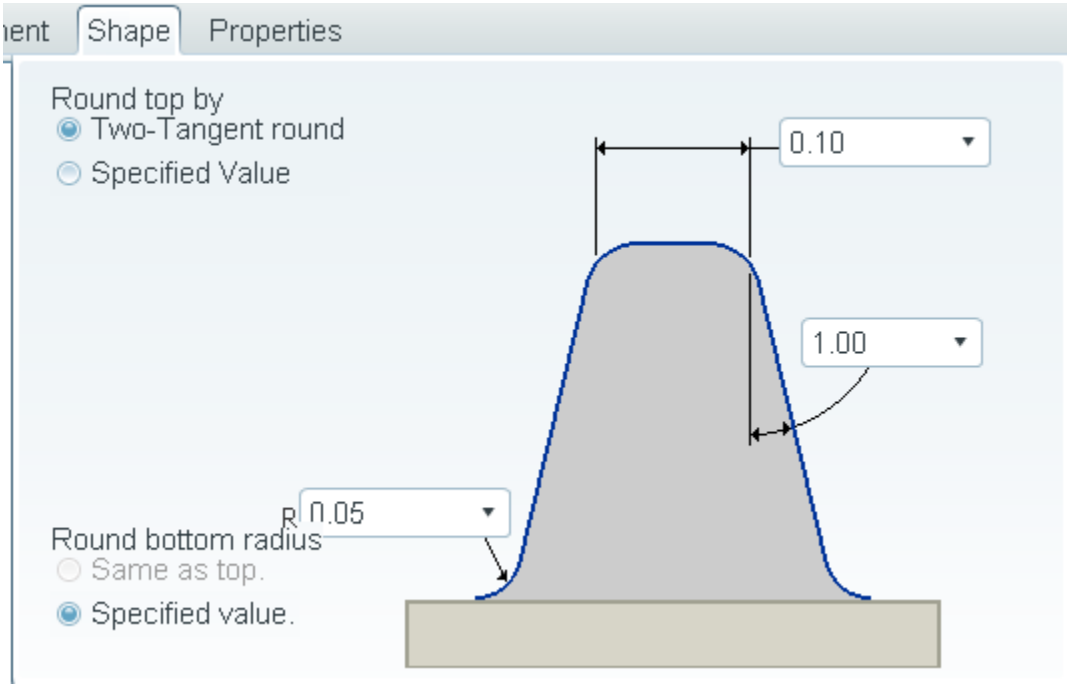


Trajectory Rib Tool

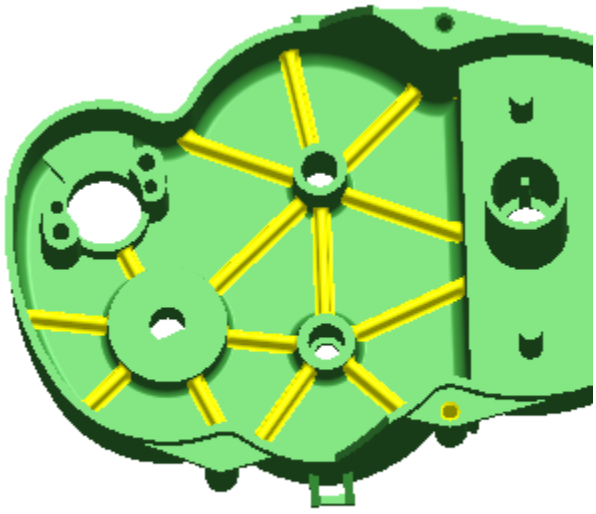
1. In Model Tree, select sketch **RIB1**, pick **Trajectory Rib** in the dashboard enter **0.1** for the thickness, select **Draft** icon, **Internal Rounds**, **External Rounds**



2. Select **Shape** tab and enter **1** for the draft value, and **0.05** for internal rounds, and **Two-Tangent round**, **MMB** or

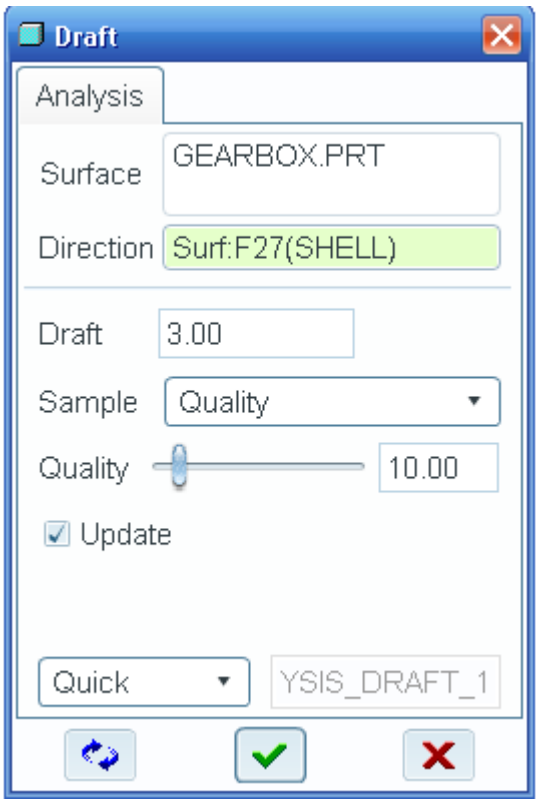


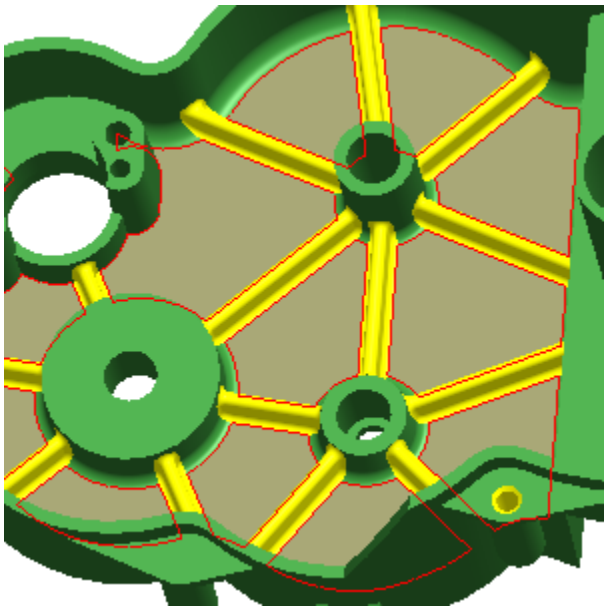
3. In Model Tree, select **Trajectory Rib 1**, **Copy** , **Paste** , in the Model Tree, select sketch **RIB2,**
4. **Paste** , in the Model Tree, select sketch **RIB3,**

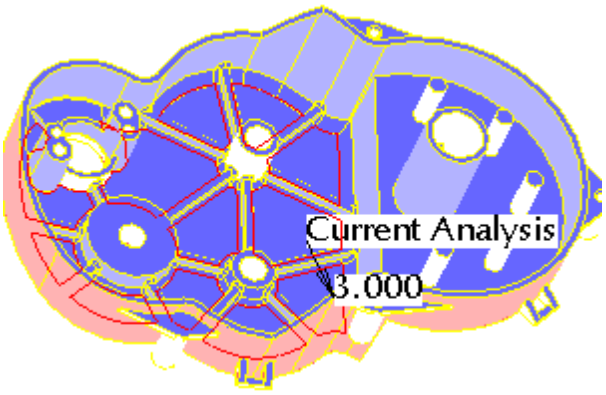


Draft Analysis

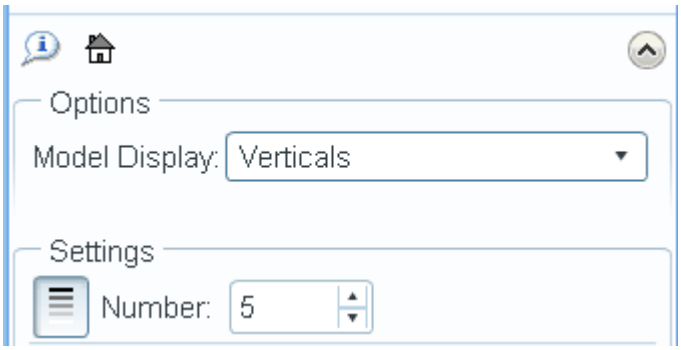
1. **Analysis, Geometry, Draft** ,select **GEARBOX.PRT** in the Model Tree, **Direction**, **Surf:F27(SHELL)**







2. At the bottom of the **Color Scale** select (options icon), **Model Display**, **Verticals**, review results,



3. **Window**, **Close**
4. **File, Erase, Not Displayed**

Sheet Metal Design & Welding

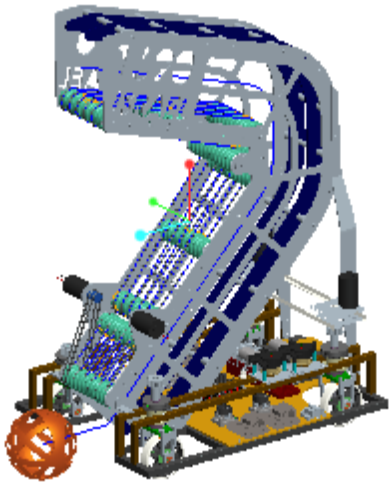
This tutorial will show how to use some of the new functionality introduced for sheet metal part design and use the new Welding user interface to connect a welded subassembly.

Sheet Metal

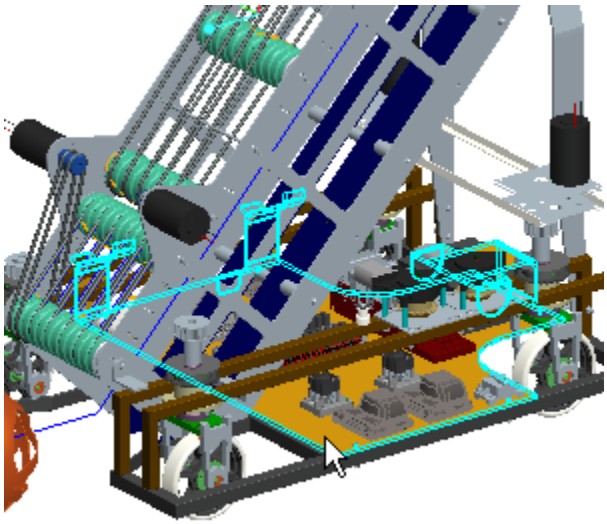
This portion of the tutorial will show the user how to pattern a flat wall, mirror selections, and apply a reinforcement form to complete a sheet metal part.

Use the Search box in the upper-right corner of the **File Open** dialog box to dynamically filter the list. This makes it much easier to find a model from a large directory.

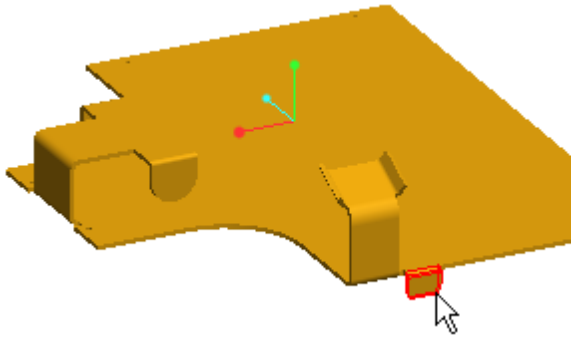
1. **File**, **Open**, **FRC-TEAM1690** folder, **frc-team1690-robot.asm**, **Open**



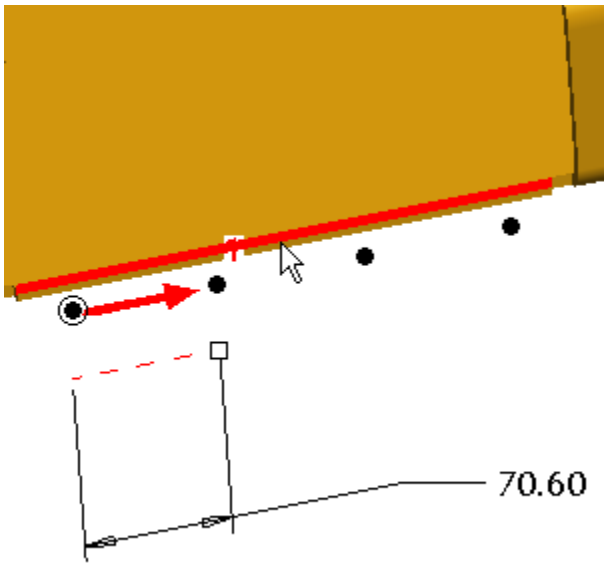
2. **LMB** pick **plate_electronics.prt** from Model Tree or graphics window, **RMB Open**



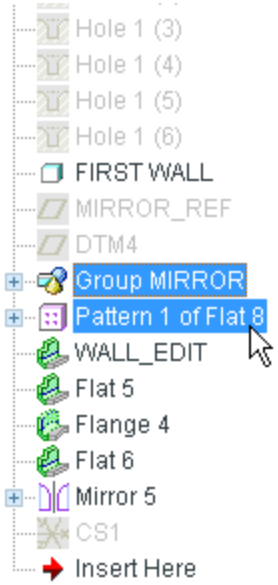
3. **LMB** pick the **Flat 8** feature as shown



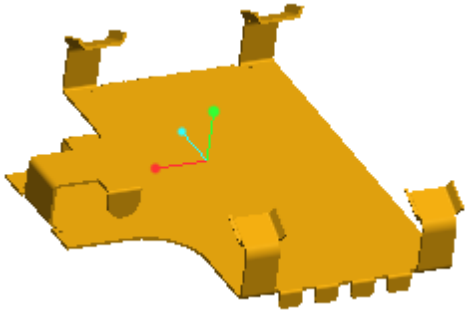
4. **Edit, Pattern**
5. **LMB** pick attachment Edge:F20(Flat_7) as the direction reference
6. Flip the pattern direction
7. change the number of pattern members to **4**
8. Set the member spacing to **70.6**



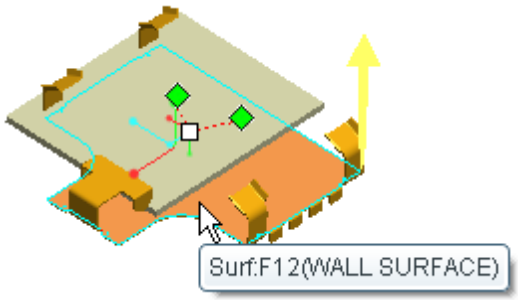
9. Select ✔ from the dashboard or **MMB**
10. **CTRL-LMB** pick **Group MIRROR** and **Pattern 1 of Flat 8** from the Model Tree



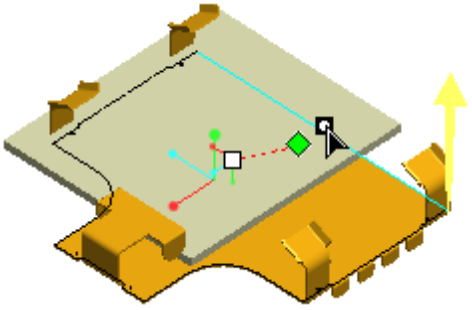
11. **Edit, Mirror**
12. **LMB** pick datum plane **MIRROR_REF** from the Model Tree as the Mirror Plane Reference
13. Select ✔ from the dashboard or **MMB**



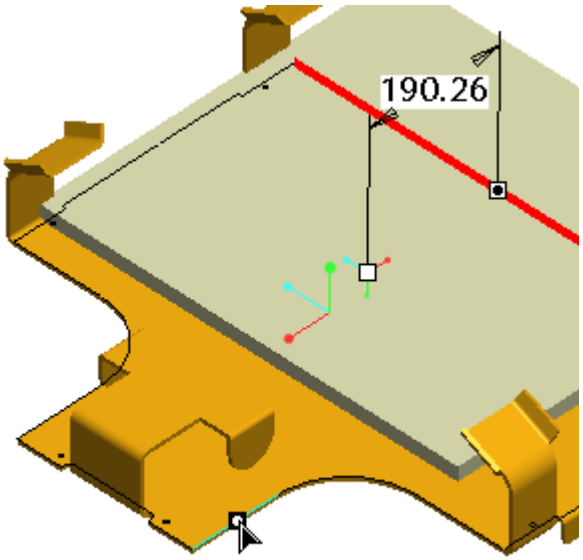
14. **Insert, Shape, Punch Form Tool**
15. **Open**, **FRC-TEAM1690** folder**, reinforcement_form.prt** and place on Surf:F12(Wall Surface)



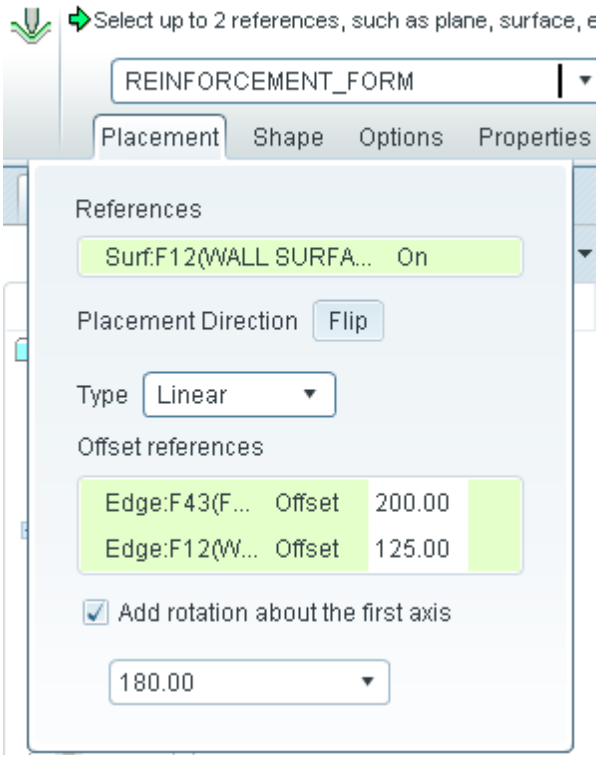
16. Drag the left most green drag handle to Edge:F43(Flat_7__2) – the top rear edge of the part

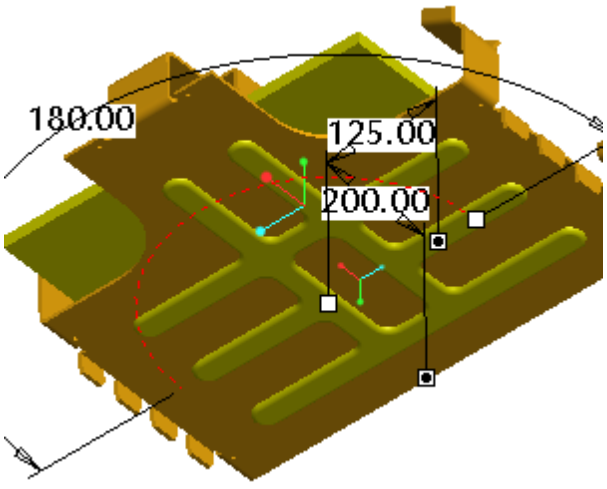


17. Drag the remaining green drag handle to Edge:F12(WALL SURFACE)

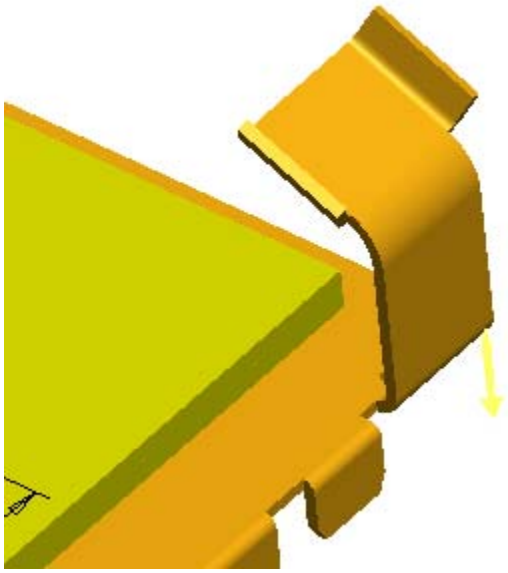


18. Select the placement tab check the **Add rotation about first axis** and enter **180** degrees
19. Enter **200** for the offset value from the first reference and **125** for the second reference

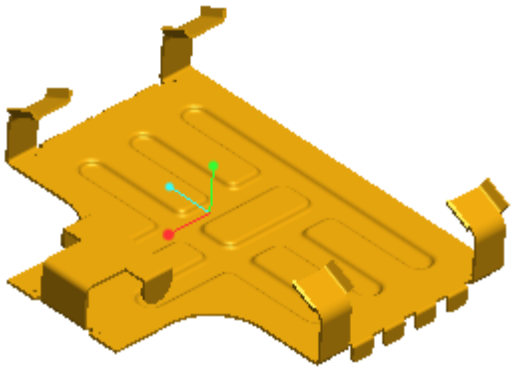




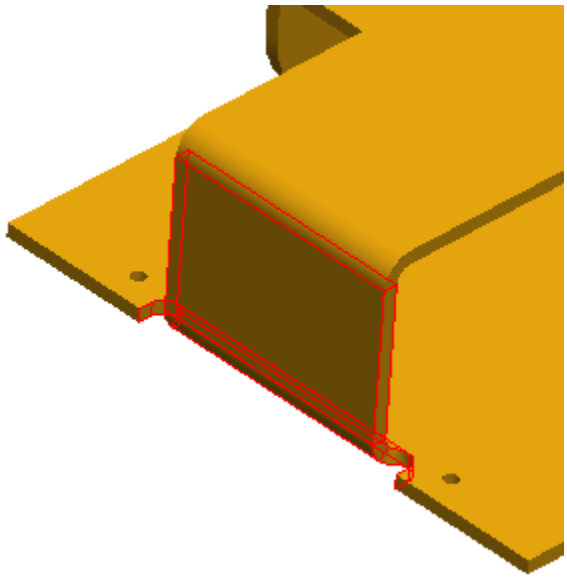
20. Ensure the yellow direction arrow is facing down or **LMB** the arrow



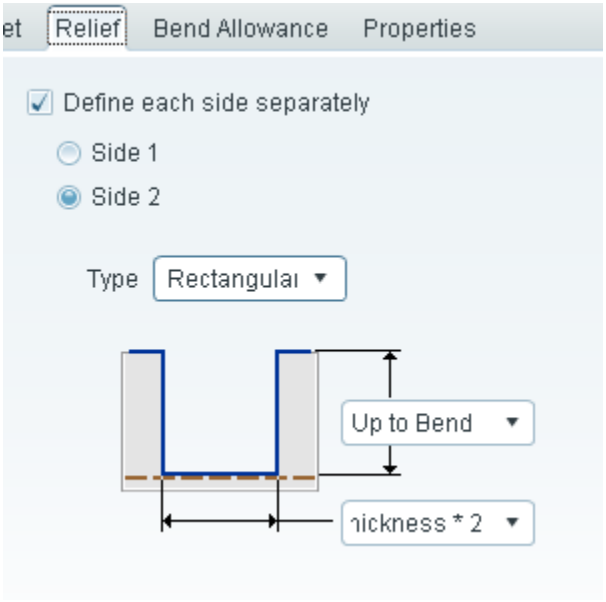
21. Select from the dashboard or **MMB**



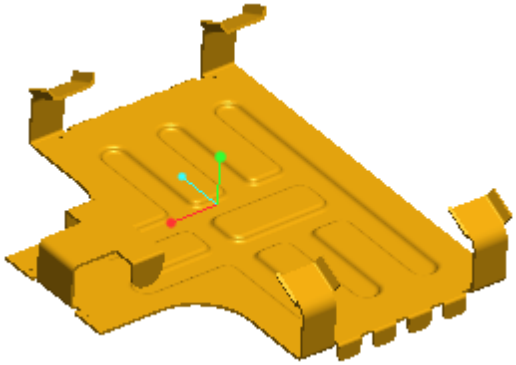
22. **LMB** pick the **WALL_EDIT** feature from the Model Tree



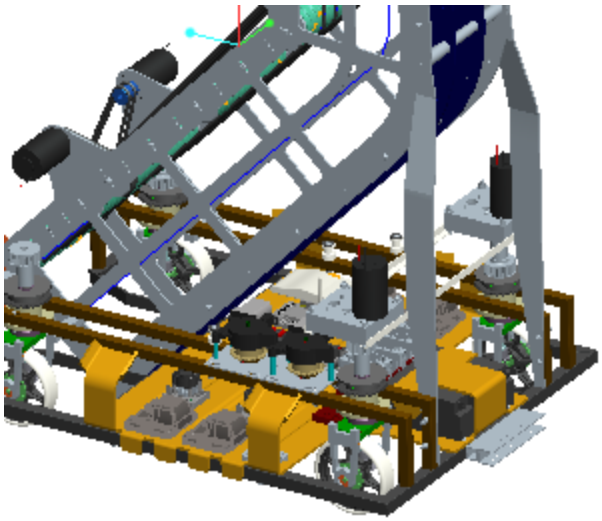
23. **RMB Edit Definition**
24. Move each of the side drag handles from the current value of 80 to 75 or type -75
25. Select the **Relief** tab from the dashboard
26. Check the **Define each side separately** box
27. Set the relief for side one as **Obround**
28. Set the relief for side two as **Rectangular**



29. Select ✔ from the dashboard or **MMB**



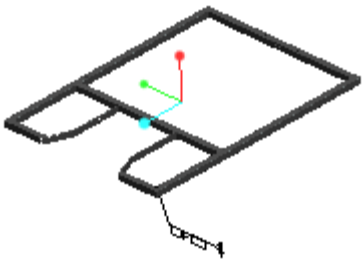
30. **Window, Close**
31. **Window, FRC-TEAM1690-Robot.asm**



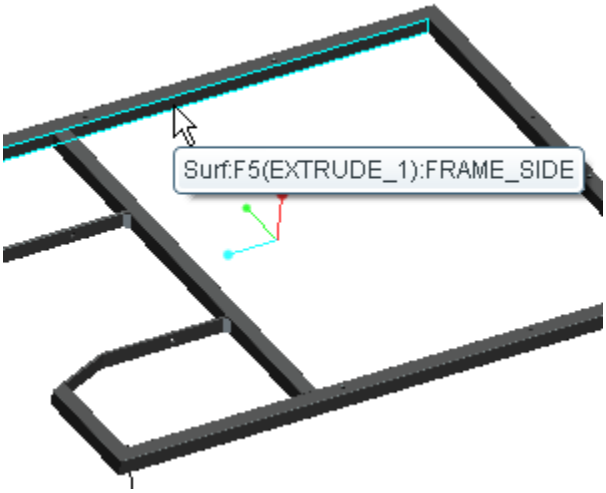
Weld

This portion of the tutorial will show the user how to leverage the new user interface to place multiple sets of fillet welds, apply material properties, combine annotations, and easily change the weld definition.

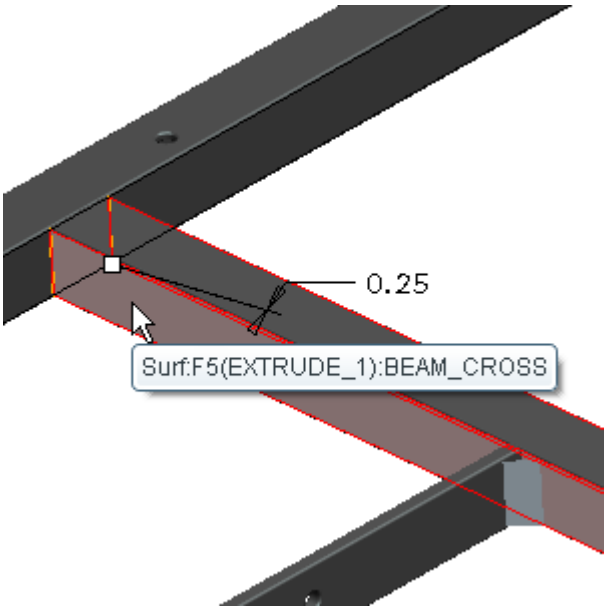
1. Expand **CHASSI.ASM** from the Model Tree, **LMB** pick **LOWER_FRAME_WELD.ASM**, **RMB Open**

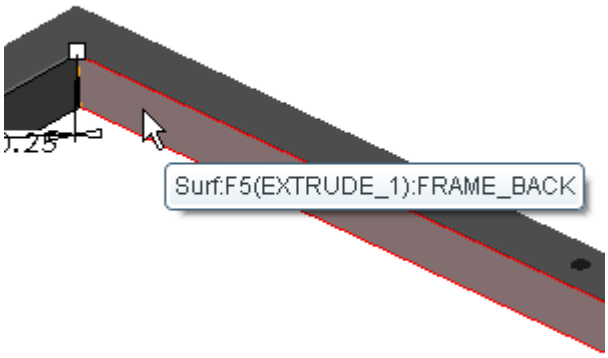


2. **Application, Welding**
3. **Insert, Weld, Fillet Weld** or select 
4. **LMB** pick **Surf:F5(EXTRUDE_1):FRAME_SIDE**

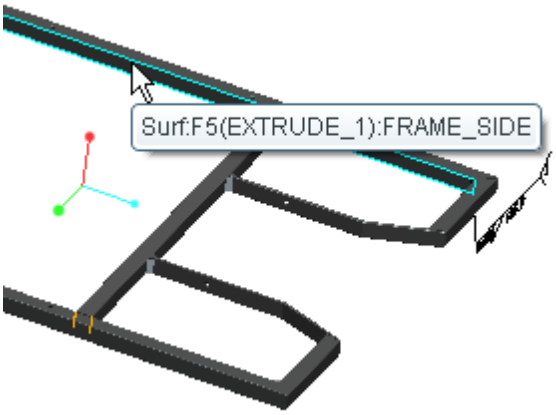


5. **RMB Side 2**
6. **LMB** pick **Surf:F5(EXTRUDE_1):BEAM_CROSS**
7. **CTRL-LMB** pick **Surf:F5(EXTRUDE_1):BEAM_CROSS –** the opposite side
8. **CTRL-LMB** pick **Surf:F5(EXTRUDE_1):FRAME_BACK**

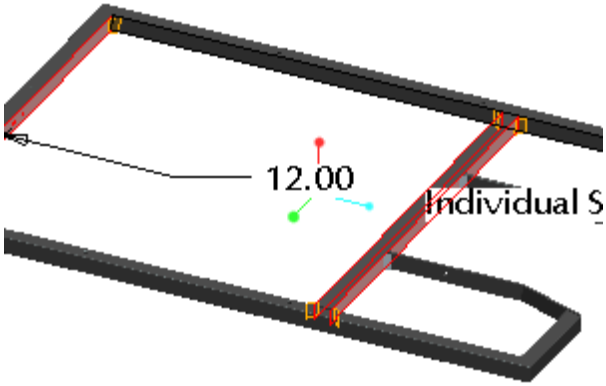




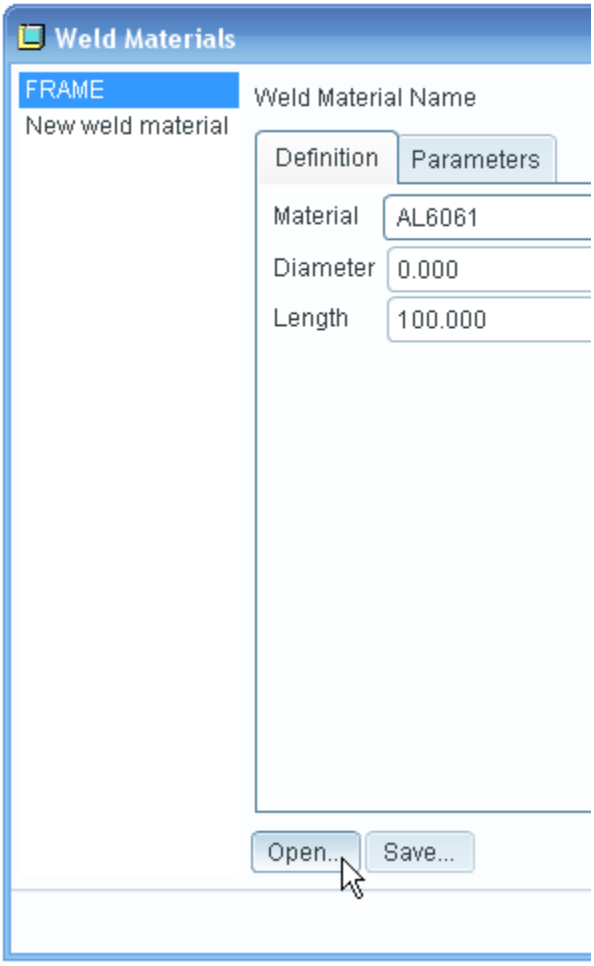
9. Change the Weld Leg Length **D** to **12**
10. **RMB New Set**
11. **LMB** pick **Surf:F5(EXTRUDE_1):BEAM_CROSS**
12. **RMB Side 1**



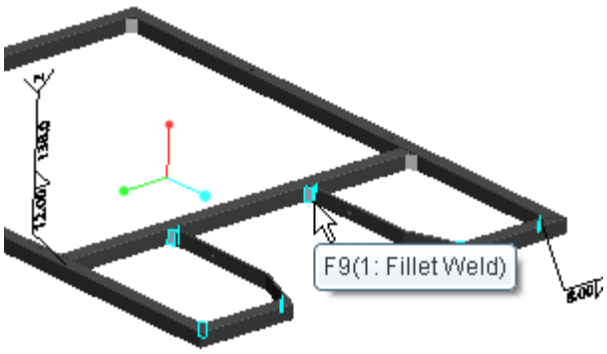
13. **LMB** pick **Surf:F5(EXTRUDE_1):BEAM_CROSS**
14. **CTRL-LMB** pick **Surf:F5(EXTRUDE_1):BEAM_CROSS –** the opposite side
15. **CTRL-LMB** pick **Surf:F5(EXTRUDE_1):FRAME_BACK**



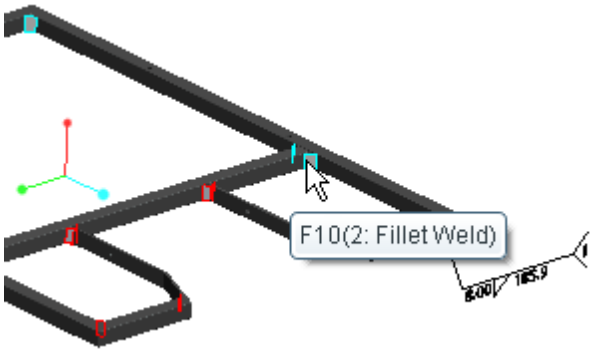
16. Select the **Options** tab and change the Weld Geometry Type to **Light**
17. Change the Weld Geometry Type back to **Surface**
18. Select **Define** for Weld Material
19. Select **Define** for Material and select **AL6061**, ⏭, **Ok**
20. Select **Open** and select **FRC-TEAM 1690** folder **, frame.spwm, OK**



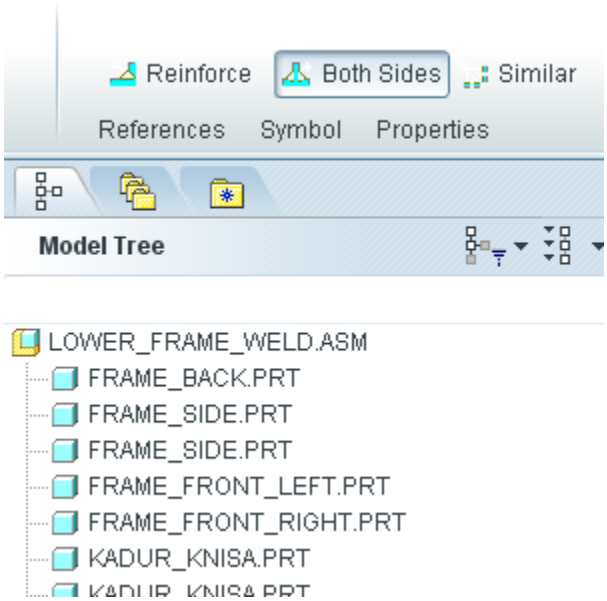
21. Select ✔ from the dashboard or **MMB**
22. Toggle on **Annotation Display**
23. **LMB** pick **F9(1:Fillet Weld)**



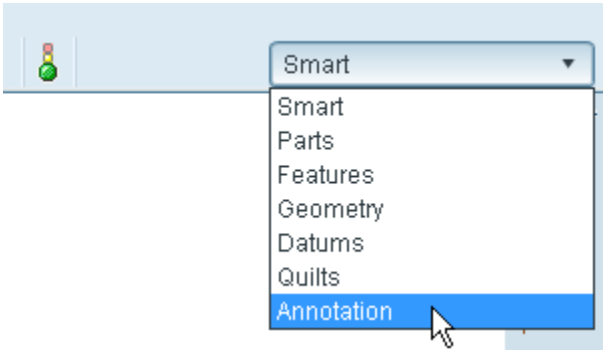
24. **CTRL-LMB** pick **F10(2:Fillet Weld)**



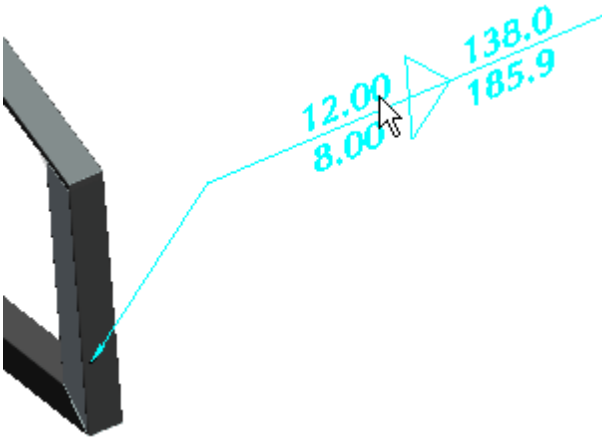
25. **Edit, Weld, Combine** or **RMB Combine** to consolidate both welds to the same annotation
26. Ensure **Both Sides** is selected



27. Select ✔ from the dashboard or **MMB**
28. Change the selection filter from **Smart** to **Annotation**



29. **LMB** pick the weld value of 12 in the annotation



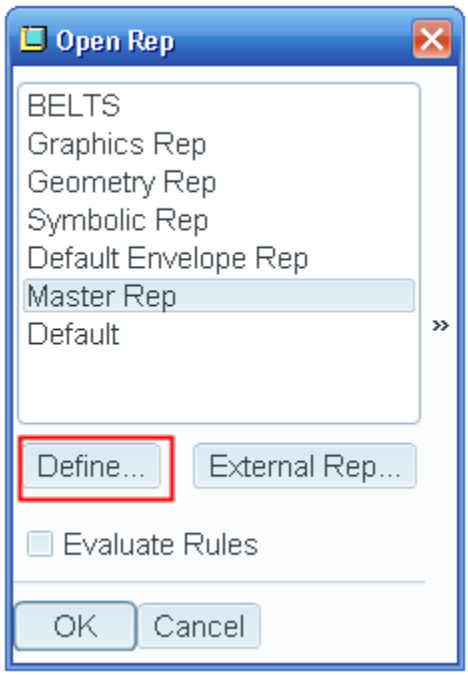
30. **RMB Value** enter 8
31. Select ✔ or **MMB**
32. **Edit, Regenerate**
33. **Window, Close**

Flexible Assembly

This tutorial will show how to create simplified representations on the fly, restructure components, copy-n-paste components to multiple locations and use the new explode animation.

Simplified Representation On-the-Fly

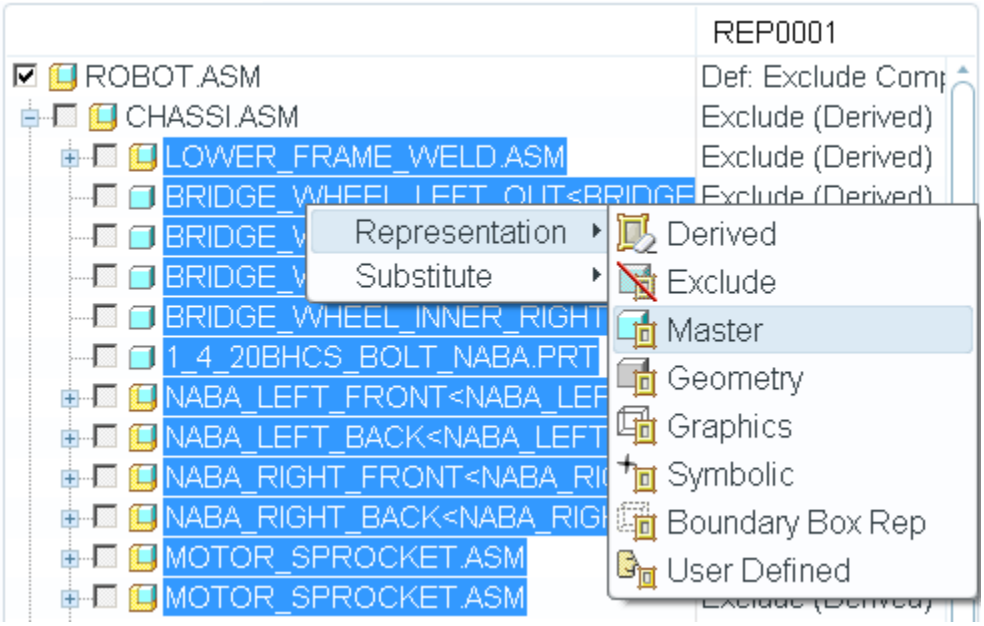
8. **File, Open**, **FRC-TEAM 1690** folder, **frc_team1690-robot.asm, Open Rep…, Define…**



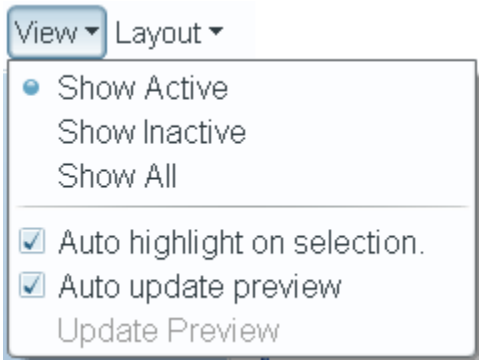
9. In the dialog box type **frame** for the simplified representation name,

> The dialog box and columns can be resized to simplify identification of desired objects. When selecting for RMB actions, pick the object name, NOT the checkbox. Selecting the checkbox will activate the default rule.

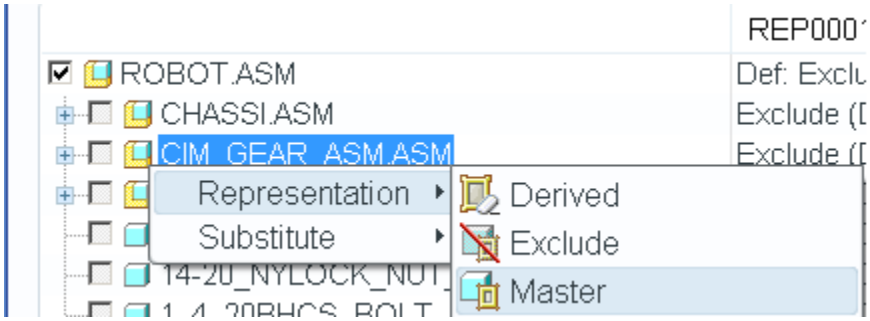
10. Expand the **CHASSI.ASM, LMB LOWER_FRAME_WELD.ASM** + Shift + first occurrence of **MOTOR_SPROCKET.ASM, RMB Representation, Master**

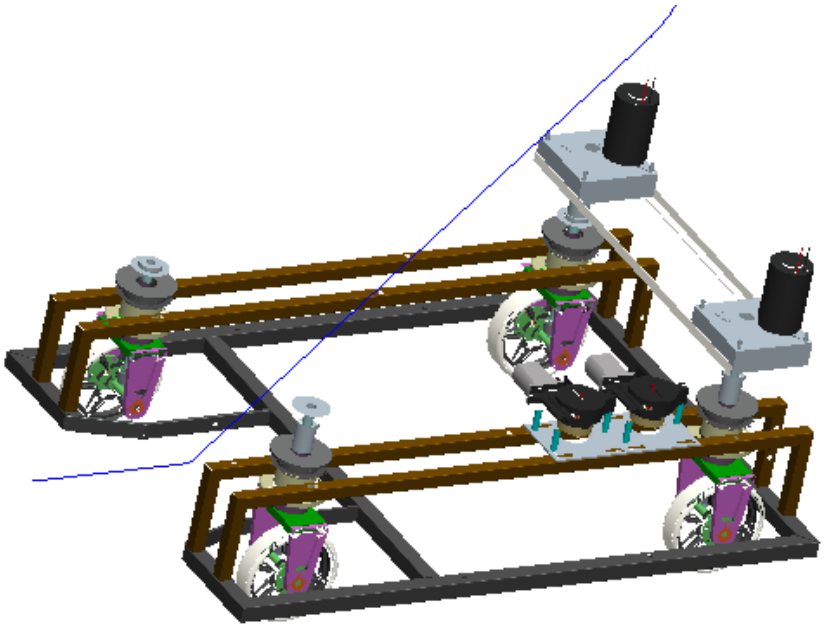


11. At the upper-right of the dialog box, try view options - **View, Show Active, Show Inactive, Show All**

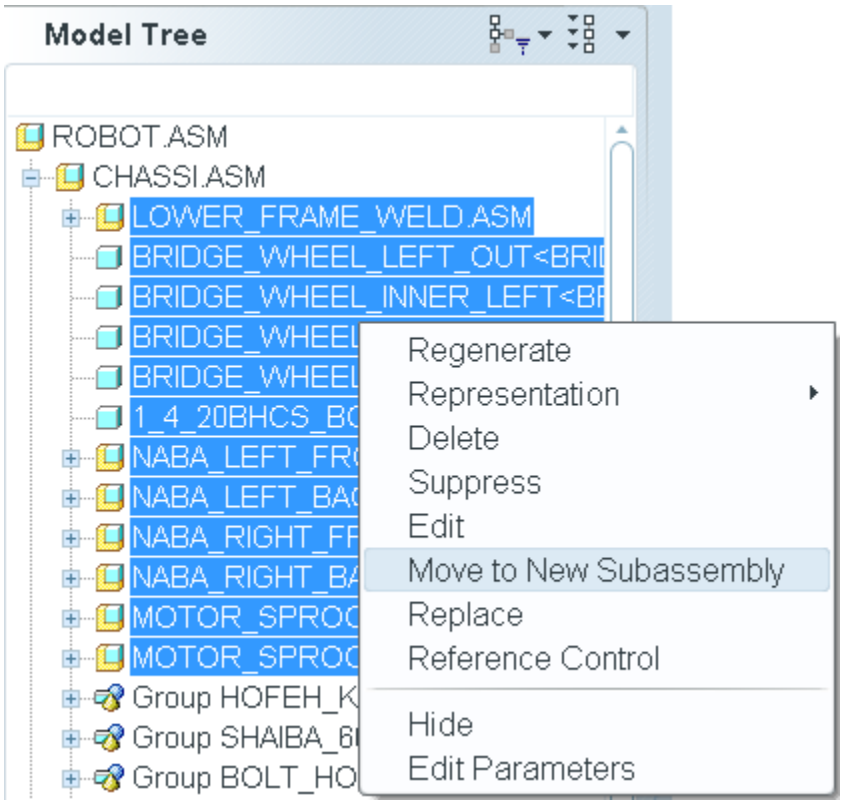


12. Select **CIM_GEAR_ASM.ASM, RMB Representation, Master, OK**

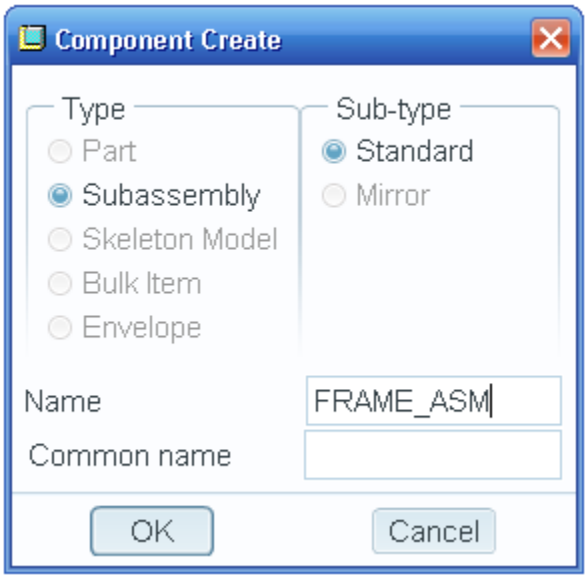




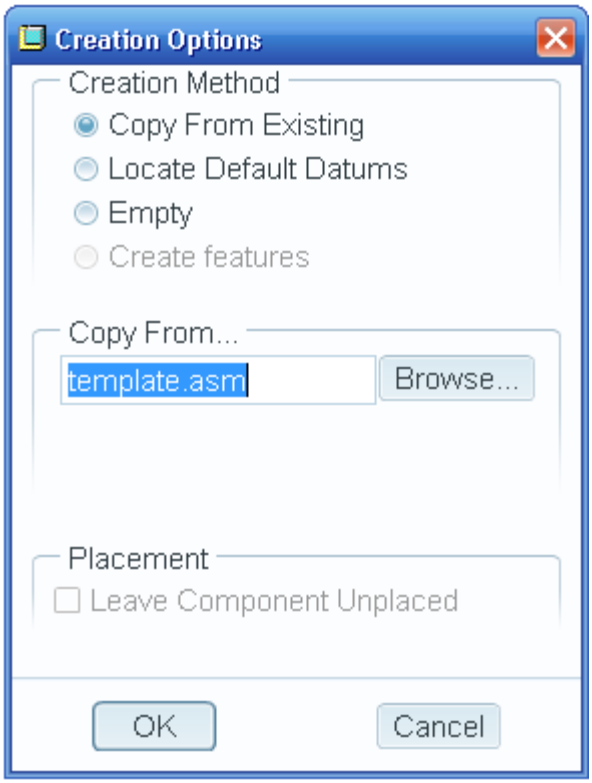
13. In Model Tree, expand **CHASSI.ASM and** select **LOWER_FRAME_WELD.ASM,** Shift + **MOTOR_SPROCKET.ASM , RMB Move to New Subassembly**



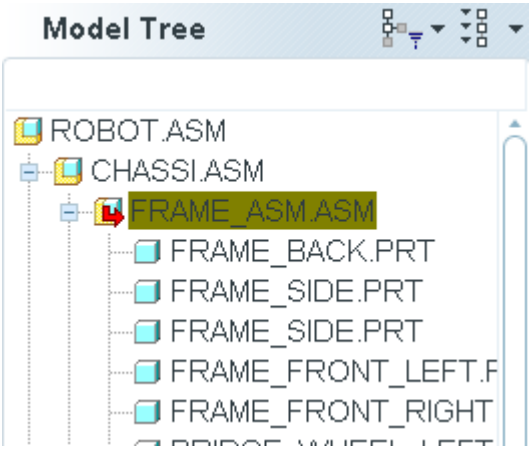
14.Type **Frame_ASM** for the name, **OK**



15.**Copy From Existing, Browse**, **template.asm, OK, RMB Default Constraint,** ✔



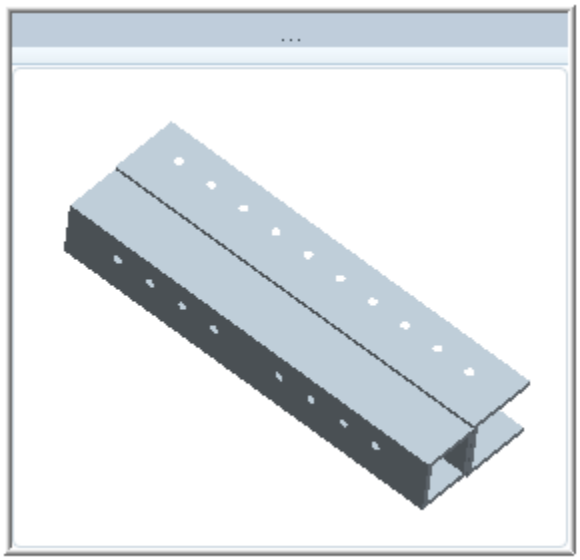
16. In the Model Tree, select and drag **CIM_GEAR_ASM.ASM** into newly created subassembly **FRAME_ASM.ASM**
17. **FRAME_ASM.ASM, RMB Open, Master Rep, OK**



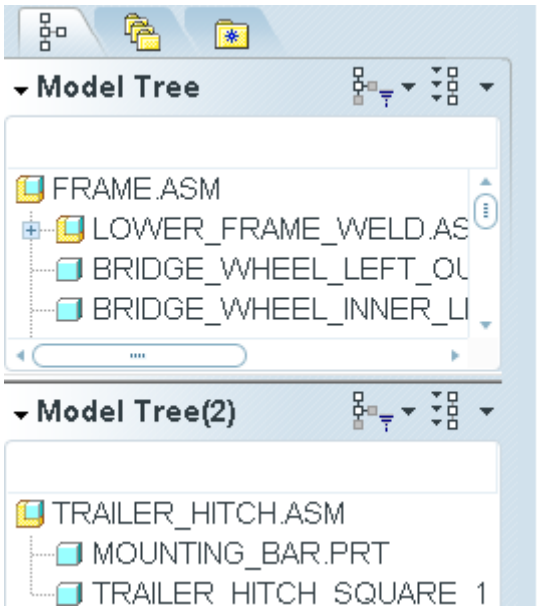
When restructuring, you must still be cognizant and careful about implications of parent/child and external references.

Assembly Enhancements

1. **Assemble** , **FRC-TEAM1690** folder**, trailer_hitch.asm, Open,** select **In Window** and **Separate Window**

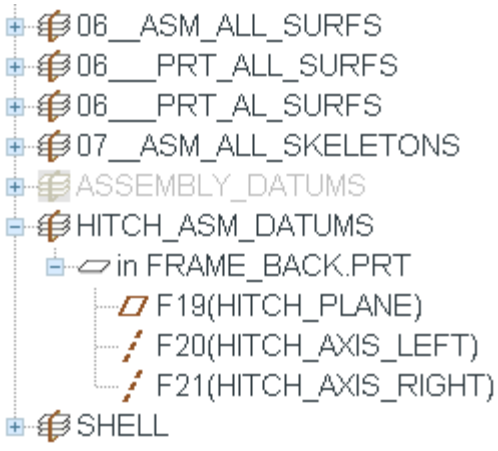


2. Model Tree(2) switch to **Layer Tree**



3. In Model Tree(2), **ASSEMBLY_DATUMS,** select **F7(HITCH_PLANE)**

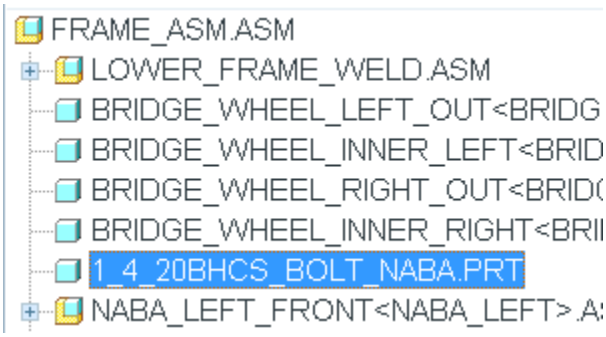
4. In the **FRAME_ASM.ASM** layer, expand **HITCH_ASM_DATUMS,** expand **FRAME_BACK.PRT, F19(HITCH_PLANE)**



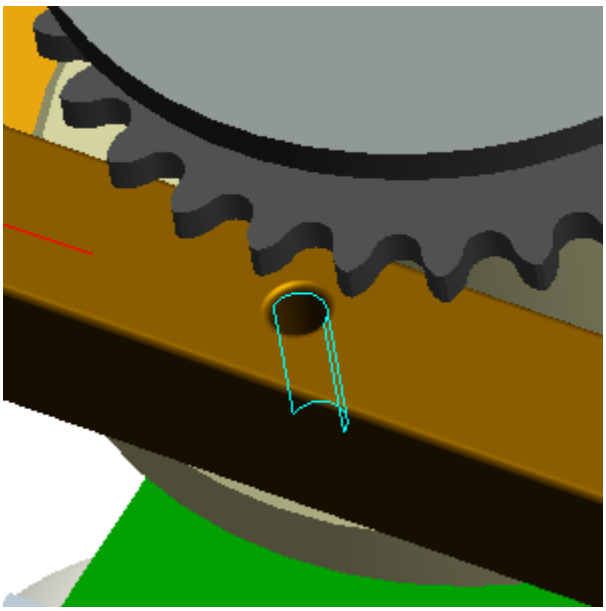
5. In the **TRAILER_HITCH.ASM layer**, select **F8(HITCH_ASM_AXIS_LEFT)**
6. In the **FRAME_ASM.ASM**, select **F20(HITCH_AXIS_LEFT),**
7. Select to return to the Model Tree

Copy and Paste with RMB to Multiple Locations

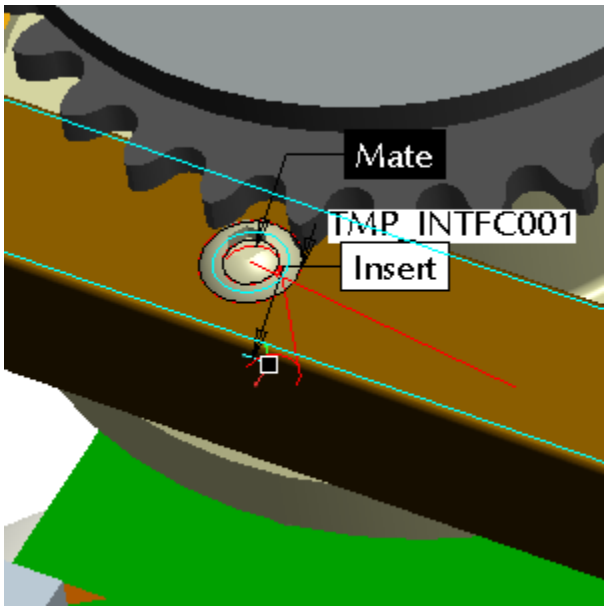
1. Select **1_4_20BHCS_BOLT_NABA.PRT, Edit**, **Copy** **, Edit, Paste**



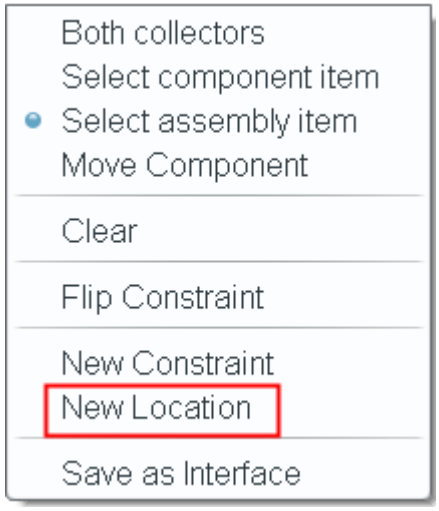
2. Select **BRIDGE_WHEEL_LEFT_OUT:Surf:F7(HOLE_1)** for the insert surface



3. Select **BRIDGE_WHEEL_LEFT_OUT:Surf:F5(EXTRUDE_1)** for the mate surface



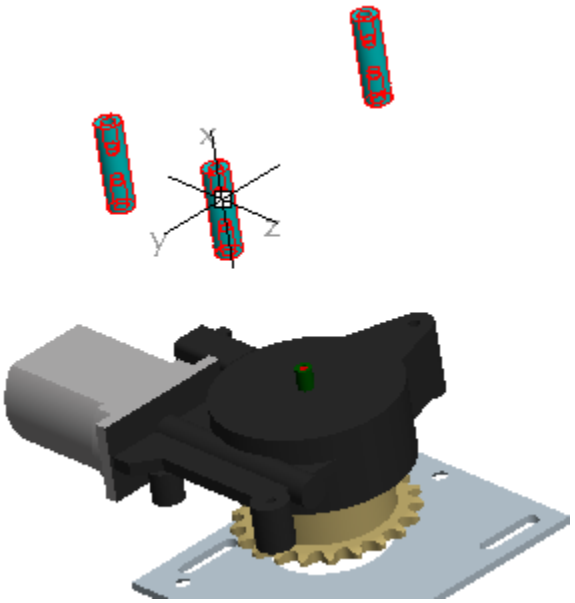
4. **RMB, New Location,** on the other side of the frame, select **BRIDGE_WHEEL_RIGHT_OUT:Surf:F7(HOLE_1), BRIDGE_WHEEL_RIGHT_OUT:Surf:F5(EXTRUDE_1),** ✔



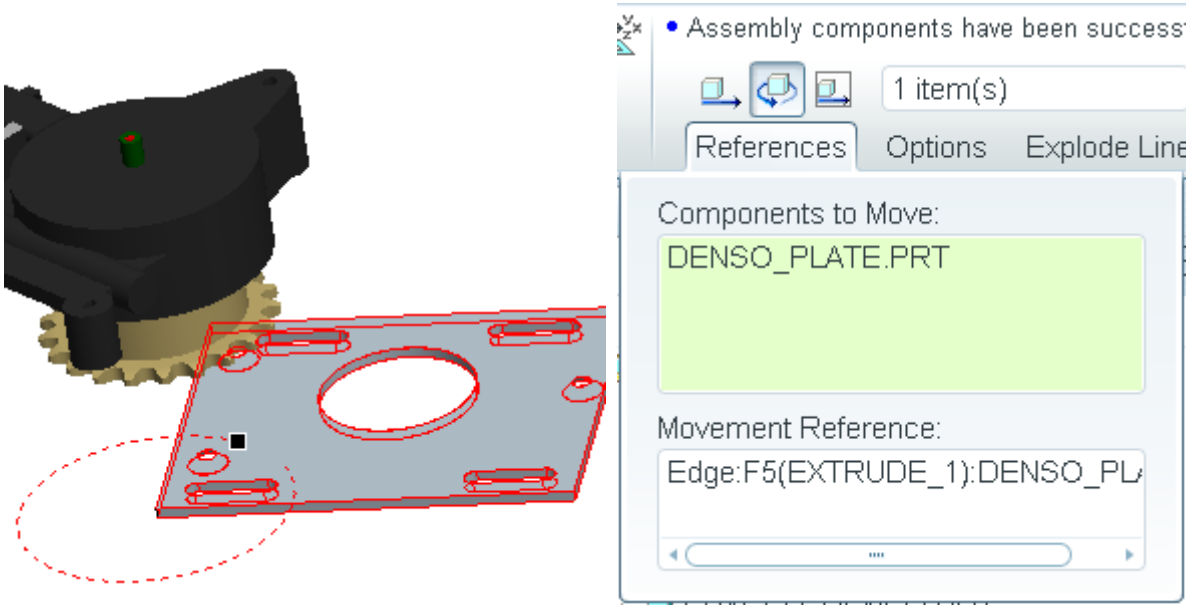
Explode Animation and Edit Position

1. Select **MOTOR_SPROCKET.ASM** in the Model Tree**, RMB Open**
2. **View Manager** , **Explode Tab,** Double-click **Default Explode, New, Enter, Properties**

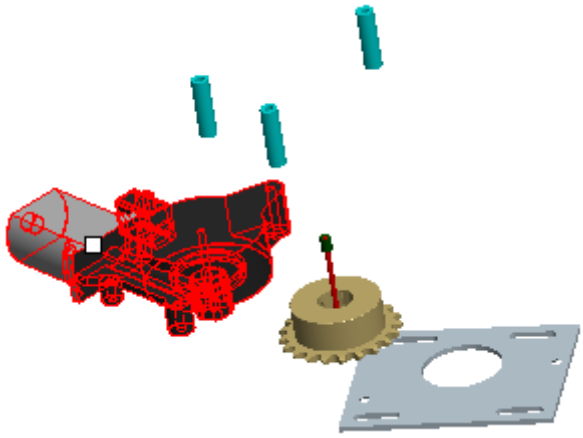
3. **Edit Position**, **Translate**, **CRTL**+ select the 3 **SPACER_DENSO.PRT,** grab and hold **X-axis,** move up as shown below



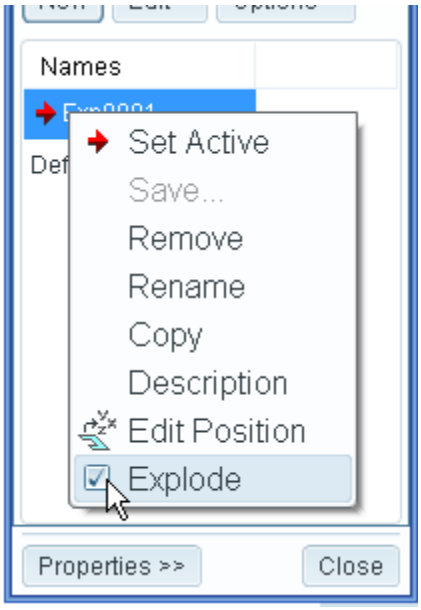
4. **Rotate**, **,** select **DENSO_PLATE.PRT**, **Edge:F5(EXTRUDE_1)**, grab and hold drag handle and move as shown below



5. **View Plane** , select **DENSO_WINDOW_MOTOR_2.PRT,** grab and hold drag handle and move as shown below,



6. **List**, **Edit, Save, OK,** Double-click **Default Explode,** Double-click **Exp0001**
7. **RMB,** Uncheck **Explode**



8. Select **Close** from dialog box
9. **Window**, **Close**
10. **File, Erase, Not Displayed**

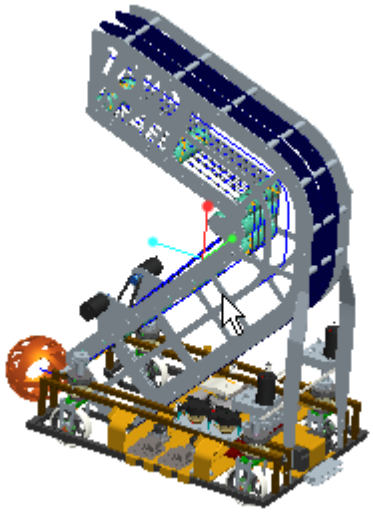
Simulation

This tutorial will show how to set up one of the new mechanism connections for enhanced machine simulation, then setup, run, and analyze the results of a structural analysis of a model using Mechanica.

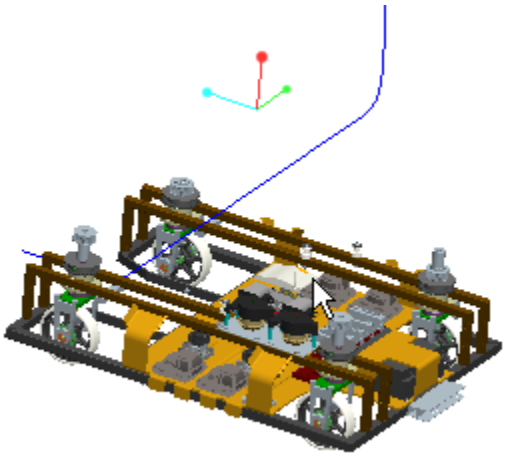
Mechanism Belt Connection

This portion of the tutorial will show the user how to set up a belt connection within mechanism mode.

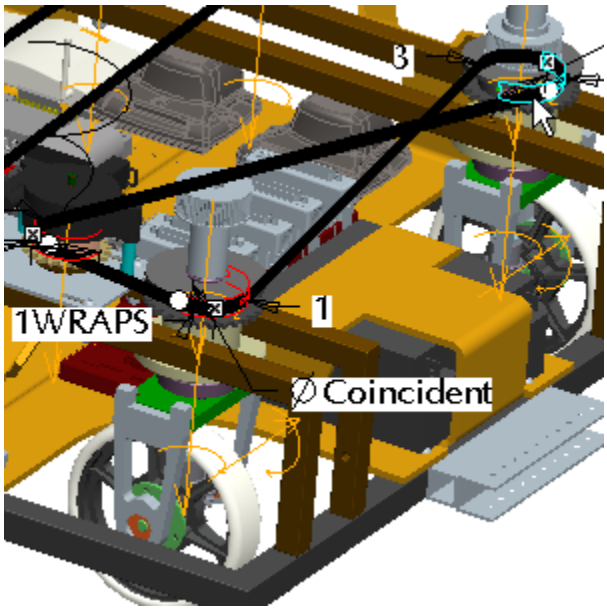
1. **File, Open**, **FRC-TEAM1690** folder, **frc-team1690-robot.asm**, **Open**



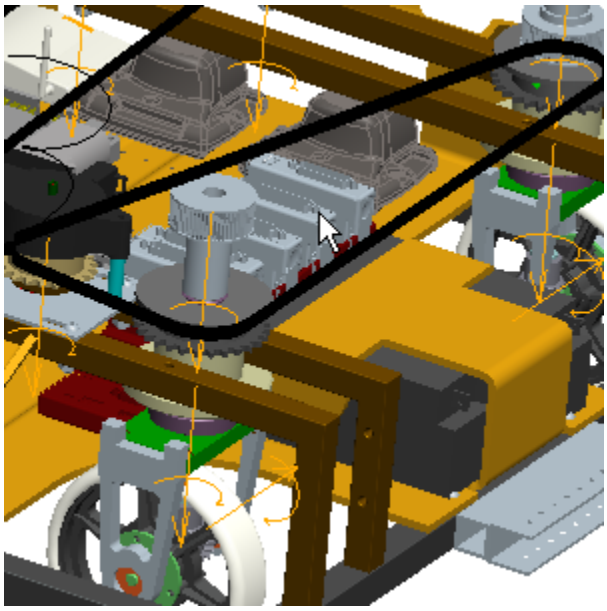
2. **View**, **View Manager** or select , double click **Belts** simplified rep, **Close**



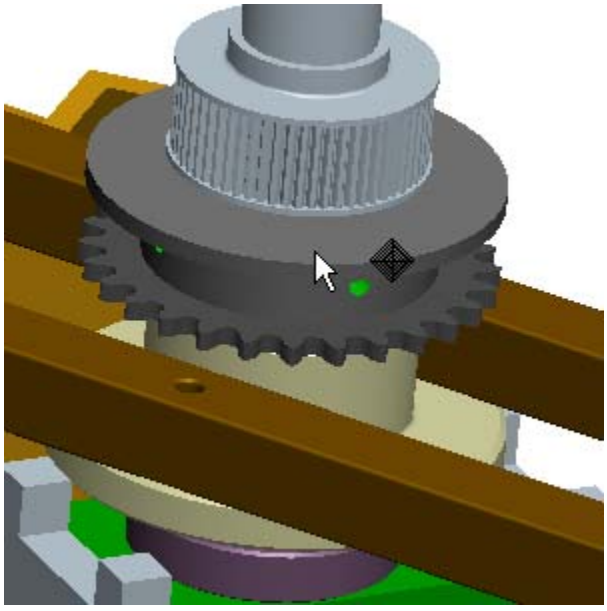
3. **Application**, **Mechanism**
4. **Insert, Belts** or select
5. **LMB** pick Surf:F2(IMPORT_FEATURE):SPROCKET__CHAIN,
   Ctrl+LMB pick Surf:F5(REVOLVE_1):MIDDLE_SPROCKET,
   Ctrl+LMB pick Surf:F5(REVOLVE_1):MIDDLE_SPROCKET



6. Click and drag the white drag handle to untwist the third pulley



7. Select ✔ from the dashboard or **MMB**
8. **View, Orientation, Drag Components** or select click and move any of the three pulleys (Observe the other pulleys moving through the belt connection)

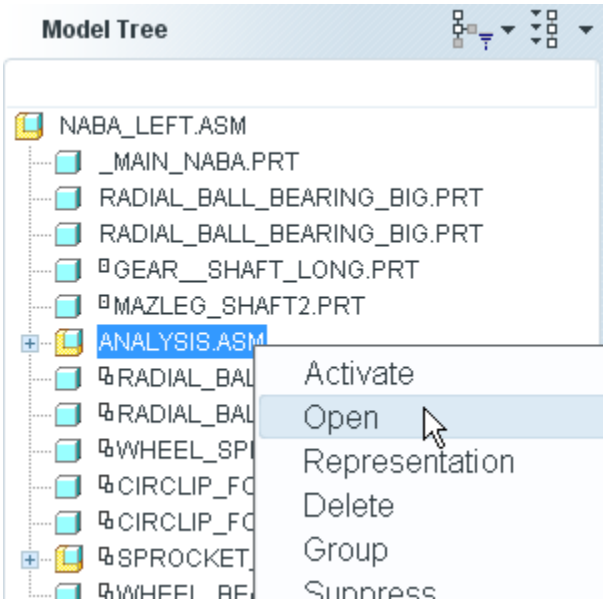


9. **MMB** three times to close the drag window
10. **Window, Close**
11. **File, Erase**, **Not Displayed**

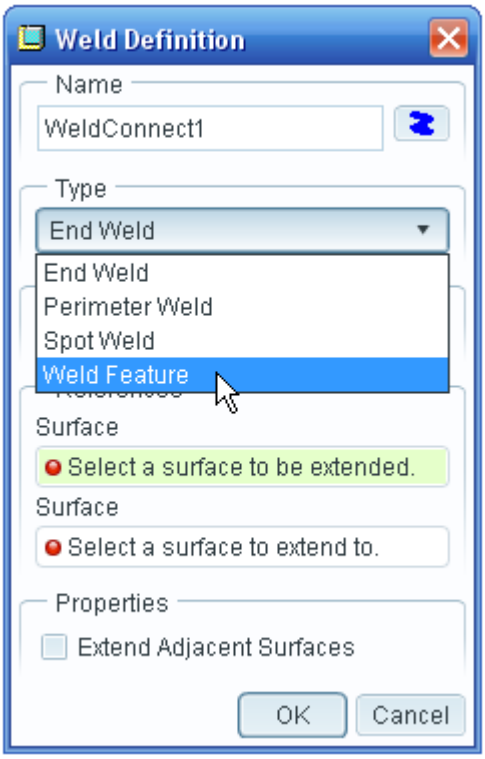
Mechanica Analysis

This portion of the tutorial will show the user how to reuse a weld feature, automatically generate mid surface shells, work with heterogeneous units, and view the results after running the analysis.

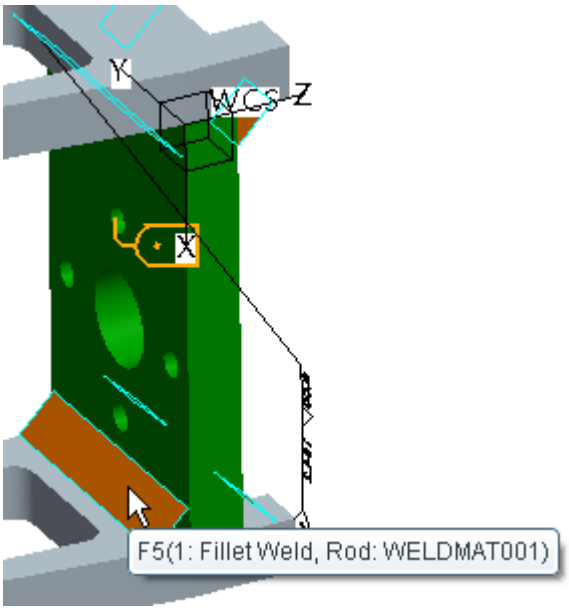
1. **File**, **Open** , **FRC-TEAM1690** folder, search with keywords **naba_left**, **naba_left.asm**, **Open** the generic
2. **Pick ANALYSIS.ASM** from the Model Tree, **RMB Open**



3. **Application**, **Mechanica**
4. **Insert, Connection, Weld** or Select
5. Select **Weld Feature** from the Type drop down menu

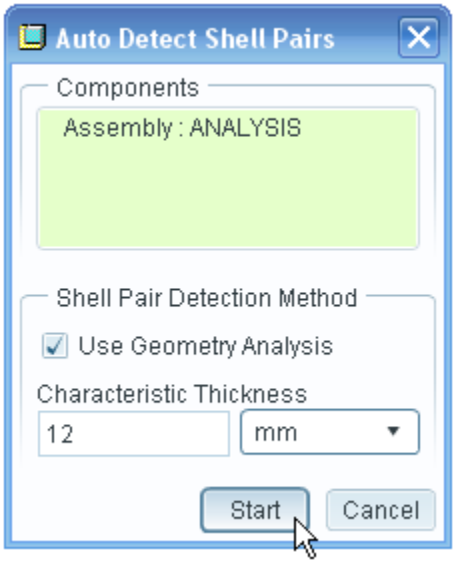


6. Select F5(1:Fillet Weld, Rod:WELDMAT001)

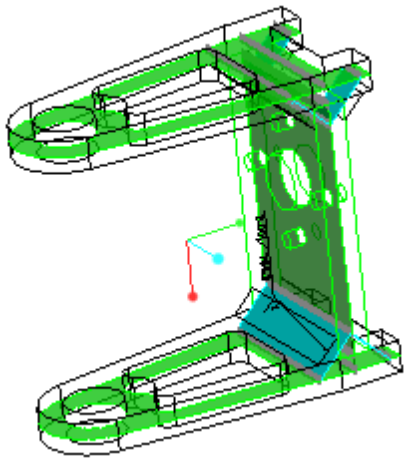


7. Select **OK**
8. **Insert, Midsurface**, **Auto Detect Shell Pairs**

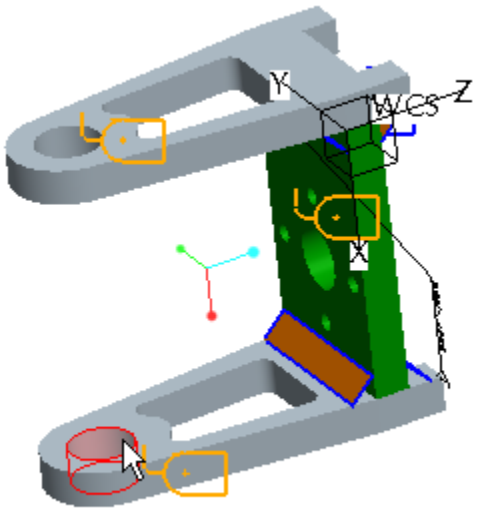
9. Select **ANALYSIS.ASM** from the Model Tree and enter **12** for the Characteristic Thickness.



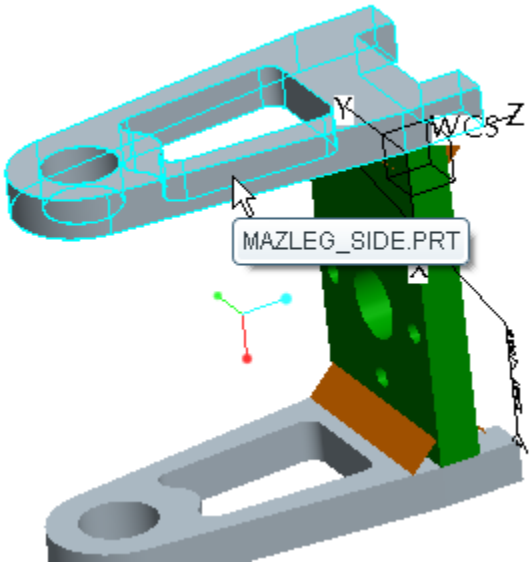
10. Select **Start**
11. **AutoGEM**, **Review Geometry**
12. Select **Apply** from the simulation geometry window



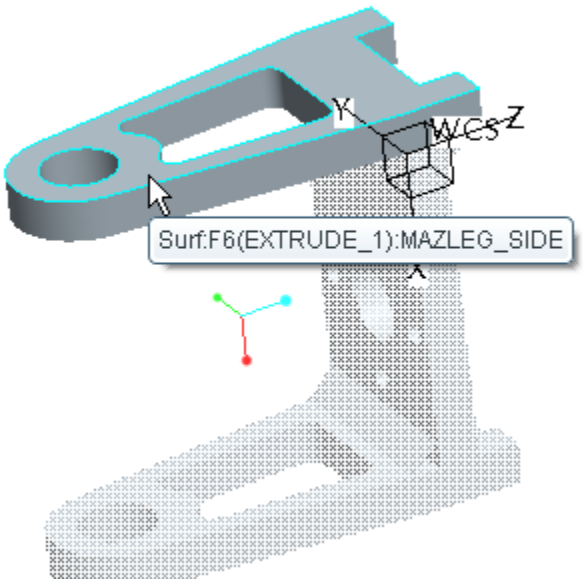
13. Select **Close**
14. **Insert**, **Pin Constraint** or select the arrow next to ▸ and select
15. **LMB** pick Surf:F6(Extrude_1):MAZLEG_SIDE



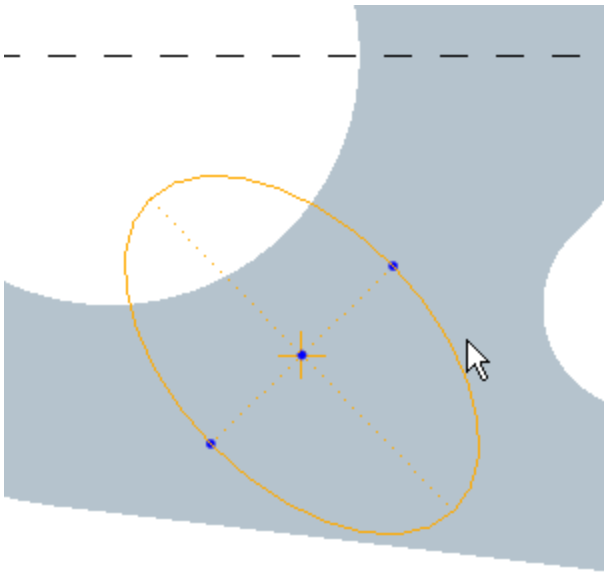
16. Select **Fixed** for both **Angular Constraint** and **Axial Constraint**
17. Select **OK**
18. **Insert, Surface Region** or select **LMB** pick part **MAZLEG_SIDE.PRT**



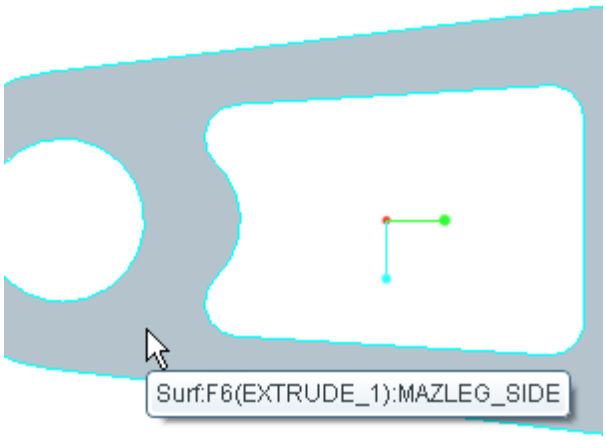
19. **RMB Define Internal Sketch**
20. **LMB** pick Surf:F6(Extrude_1):MAZLEG_SIDE



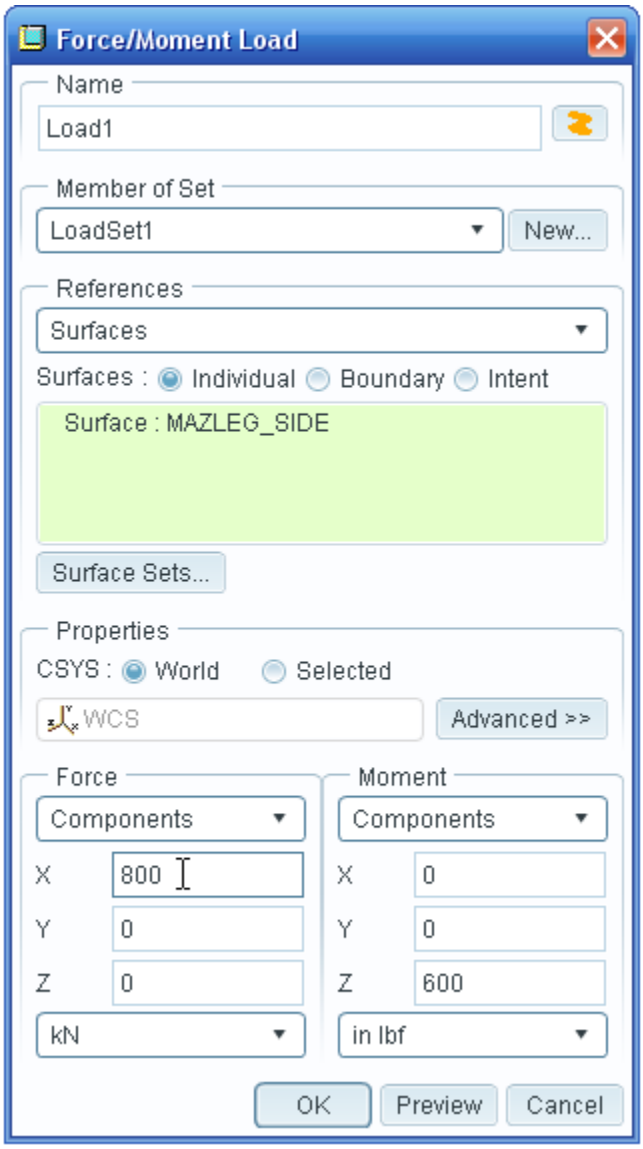
21. Click **Sketch**
22. **Sketch, Circle, Center and Axis Ellipse** or select the arrow next to and select
23. Click with the **LMB** once to define the center and a second time to define the radius and a third time to finish (This ellipse can be approximate)



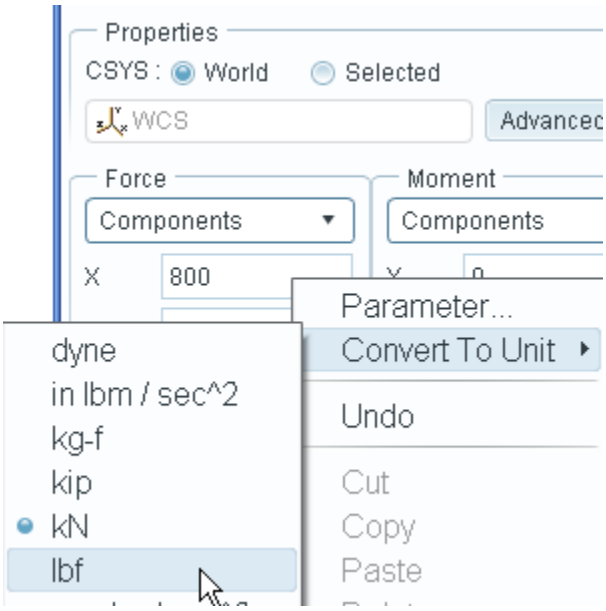
24.
25. Select Surf:F6(EXTRUDE_1):MAZLEG_SIDE as the placement surface



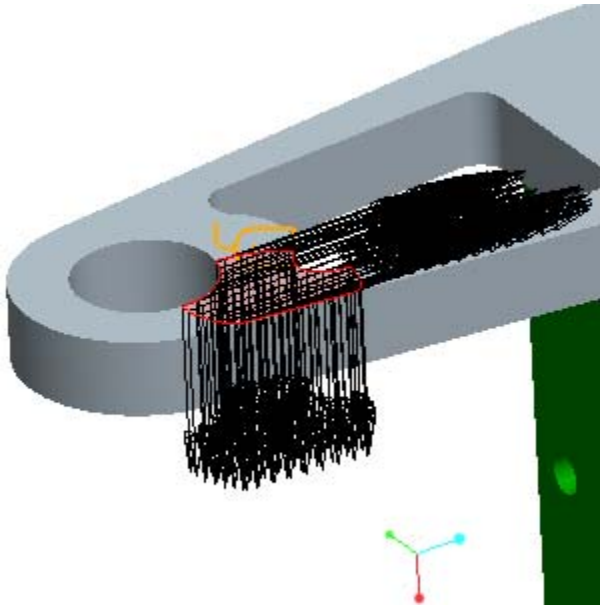
26. Select ✔ from the dashboard or **MMB**
27. **Insert, Force/Moment Load** or select , select the previously created surface region as the reference
28. Add a force of **800 N** in the X direction and add a moment of **600 in lbf** in the Z direction
29. Change the units for force to KN
30. Select the 800 Value



31. **RMB Convert to Unit, lbf**



32.Select **Preview**

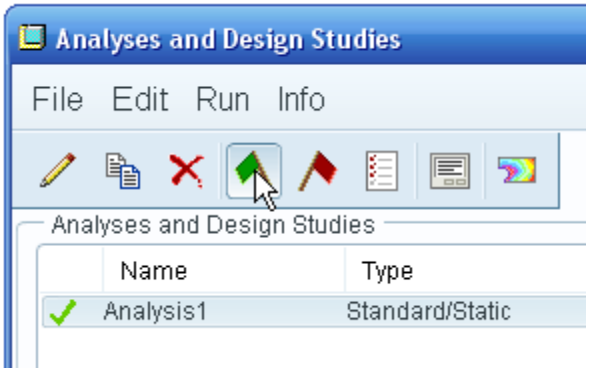


33.Select **Ok**

34.**Analysis, Mechanica Analysis/Studies**

35.**File, New Static**

36.Select **OK**

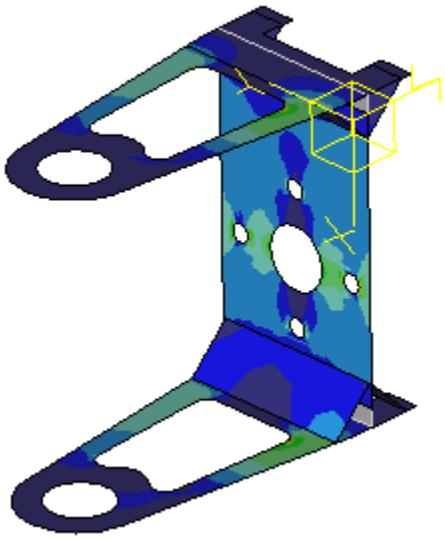


37.Select **Run** Select **No** for interactive Diagnostics

38.Once analysis has reached the complete status, **Analysis, Results** or select

39.Select **Fringe** for display type, and select **Stress** for the quantity

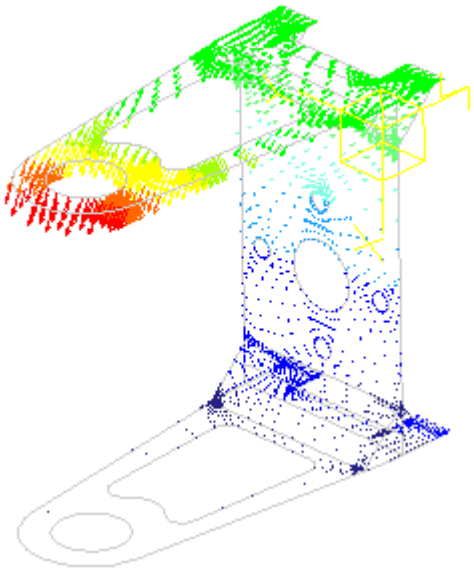
40.Select **OK and Show**



41.**Insert**, **Results Window,** select **Analysis1, OK**

42. Select **Vector** for display type and set the quantity to **Displacement**

43.Select **Ok and Show**



44. **Window**, **Swap**
45. **Window**, **Close**
46. **File, Erase, Not Displayed**

Drawing Workflow and Efficiency

Pro/ENGINEER includes many new enhancements for creating and working with 2D drawings and improved interaction with 3D drawings.

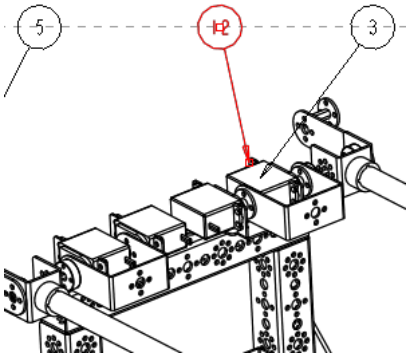
- Improved creation and display of 2D & 3D annotations
- Improved display and management of annotations
- Easy creation and manipulation of geometry annotation
- Enhanced user experience and productivity
- Enhanced capabilities and support for 2D documentation
- Enhanced capabilities and support for 2D print and plotting

Task-Based User Interface

1. **File, Open**, **FTC-ROBOT** folder, search with keywords **ftc**, **ftc-robot.drw**, **Open**

> Now drawing commands are re-organized into a ribbon-style user interface. The new user interface (UI) is designed to display only those drawing commands which are appropriate for the current task.

2. Select **Table** tab, **Annotate**, **Sketch**, **Review**, **Publish** and review the ribbon-style top level UI
3. Select **Layout** tab, hold **Alt** and select the balloon in the drawing



Publish

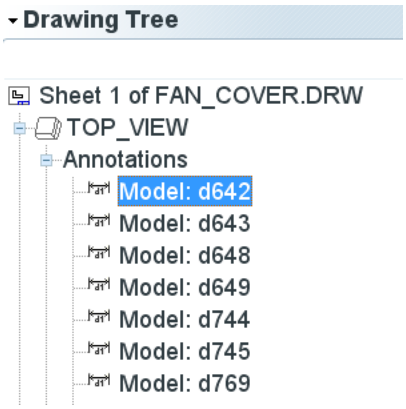
The print preview display considers the current printer configuration to determine line weights and styles, priorities and colors. The preview displays white background paper space and users have full control leveraging pan and zoom to assess preview display in the graphics window

1. Select **Publish** tab, click **Preview**, zoom in the plot preview to review what the printed output looks like before sending it to the printer, **Close Preview**
2. Check **PDF** option, **Settings** , check **Solid Hidden Lines** in Line style column, **OK**, **Preview**, in PDF reader, toggle on **Pages** , select page 1, page 2 and review the pages, open **Bookmarks** , select new_view_1, new_view_3 in the bookmarks list, review the details, **Close** Adobe Acrobat Reader
3. **Window**, **Close**

Model Annotation Tool

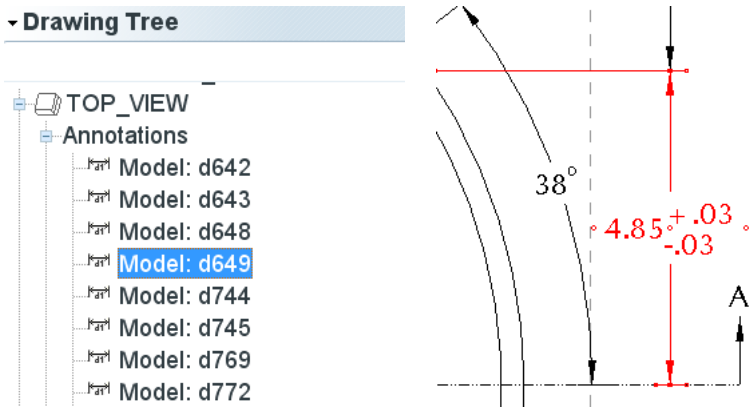
Selectable drawing objects appear in a tree hierarchy, the content of the Drawing Tree varies depending on the tab selected, simplifying the tree structure. Objects are highlighted in the graphics window when you select them from the graphics window or the Drawing Tree

1. **File, Open** , **DETAILING** folder, **fan_cover.drw**, **Open**
2. **Annotate**, expand **Annotations** of **TOP_VIEW** in the Drawing Tree, select **Model:d642, Model:d643**, review the corresponding highlighted dimensions in the graphics window



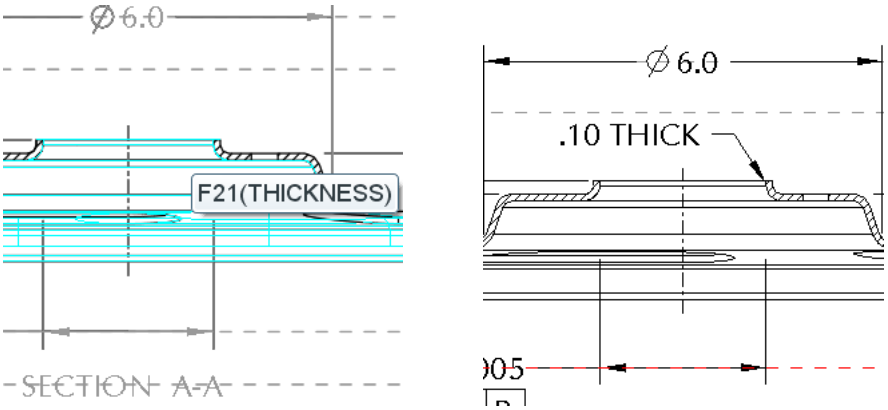
The new options gives user more control over the tolerance display and allow the user to select the dimensions true significant digits

3. **RMB** the **Model:d649** dim in the Drawing Tree, **Properties**, change **Decimal Places** to 2, **Enter** to preview update, check the **Rounded Dimension Value** option, change **Tolerance mode** from **Nominal** to **Plus-Minus**, **OK**, review the updates of the dimension

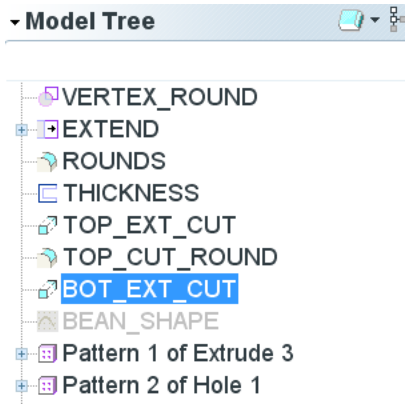


New Show Model Annotations tool for dimensions, GTOL's, notes, surface finish, symbols and datums can select by view or by feature within a view, and annotations available to be shown will preview, you just select them to show in the drawing

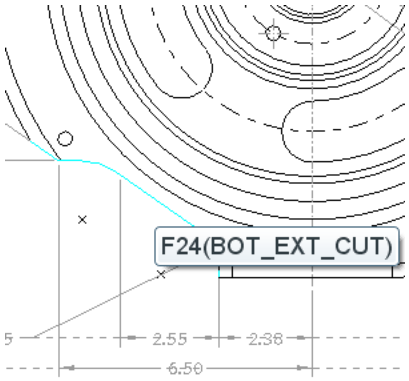
4. Select **Show Model Annotations** icon on the ribbon, select the thickness in the Front Cross Section view as shown, preview dimensions, check d891 which stands for the thickness of the part, **OK**, select the .10THICK and move it to empty area



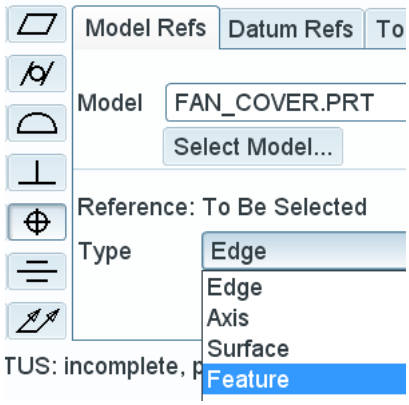
5. Select **Show Model Annotations** icon, select the **BOT_EXT_CUT** feature from the Model Tree, preview the dimensions showing up, **Cancel**



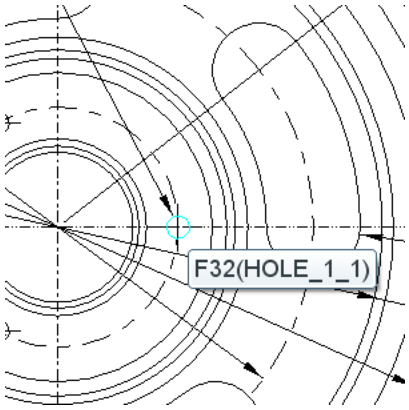
6. Select **Show Model Annotations** icon, pick model edge as shown below, see the difference in what dimensions appear, **Cancel**



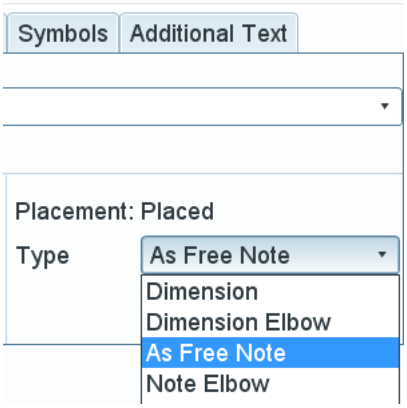
7. Select **Geometry Tolerance** from the **Insert** ribbon, select ⌖, Reference Type: **Feature**



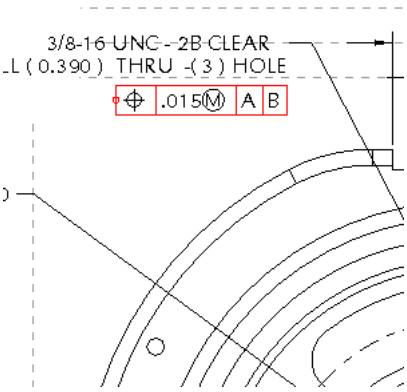
8. Select the hole feature in the Top view



9. Set **Placement Type** to **As Free Note**



10. **LMB** place the Geometry Tolerance Annotation below the hole note, **OK**



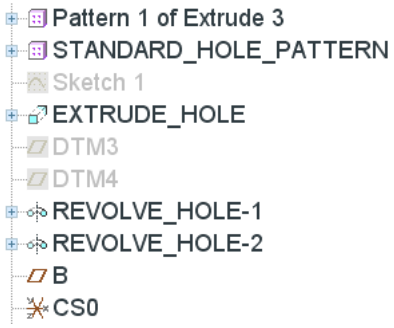
Sheet Tab & Hole Table

Drawing sheets appear as tabs across the bottom of the graphics window. The new Hole Table automatically includes extrude and revolve cuts in the table

1. Click the **hole_table** tab at the bottom of the drawing area, **RMB** on **hole_table** tab to show all shortcut options
2. Review the Hole table

	Hole Chart	new_view_17	
Hole No.	X	Y	Ø
A1	-6.84	3.64	0.38
A2	-6.84	-3.64	0.38
A3	6.33	4.43	0.38
A4	6.84	-3.64	0.38
B1	-2.00	-0.00	3/8-16 UNC
B2	1.00	-1.73	3/8-16 UNC
B3	1.00	1.73	3/8-16 UNC

3. Highlight **STANDARD_HOLE_PATTERN**, **EXTRUDE_HOLE**, **REVOLVE_HOLE-1** and **REVOLVE_HOLE-2** features in the Model Tree, and in the drawing review the corresponding highlighted holes created in different methods



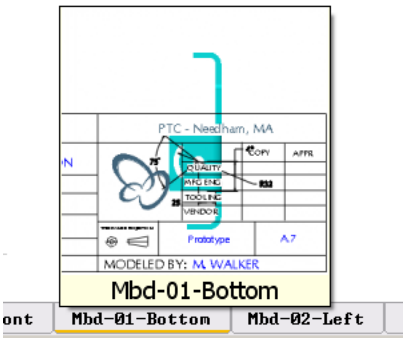
4. **Window**, **Close**

Combined View Tab

1. **File, Open**, **DETAILING** folder, **bracket.prt**, **Open**, toggle on **Annotation Display** if necessary

> You can easily navigate between the combined states of a model without opening the View Manager. Combined or All states appear as tabs, each with a thumbnail preview, in the graphics window

2. Select **View Manager**, **All**, check **Display combined views**, **Close**
3. Move cursor to combined view tabs at the bottom of the graphics window, show the thumbnail preview of combined view

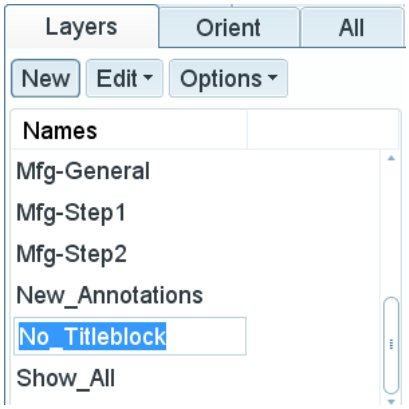


4. Click Mbd-00-Front tab, Mbd-01-Bottom, Mbd-02-Left, Mbd-03-Right, Show_All combined view tabs to review the model information from different angles, **RMB** on Show_All view tab, preview all shortcut options, select **Redefine**, change **Orientation** to **3D-Detail**, ✔

Layer Visibility

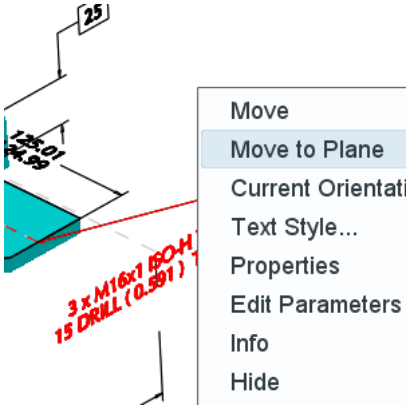
You can create layer visibility states from the View Manager, and you can toggle the display of all layer-assigned content.

1. Select , select **View Manager** , **Layers**, double-click **Mbd-00-Front**, **Mbd-01-Bottom**, **Mbd-02-Left**, **Mbd-03-Right**, **Show_All**, and review the corresponding layers visibility changes in the Layer Tree
2. **RMB** on **ANNOT_ALL_TBLOCK** in the Layer Tree on the Navigator, **Hide**, **Repaint**
3. Create **New** Layer on View Manager, **No** to Modified State Save, input the name No_Titleblock, double click **Show_All**, then double click **No_Titleblock**, review the layer state changes

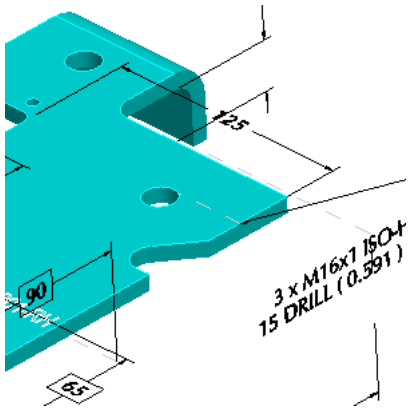


Move Annotation to Plane

1. Double click the Show_All layer state, change the global filter to **Annotation**, select the hole annotation at top right corner, **RMB** in the graphics window, **Move to Plane**

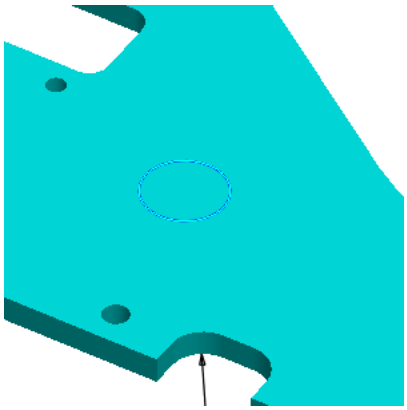


2. Select the top surface as showing, review the Z orientation update of the 3D annotation

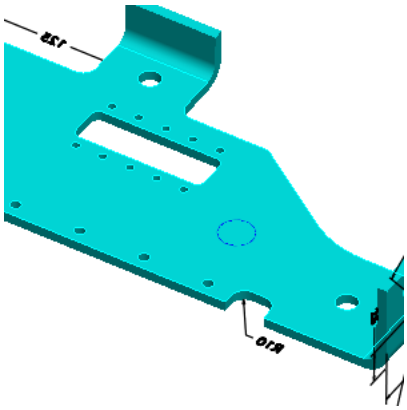


Target Datum Annotation

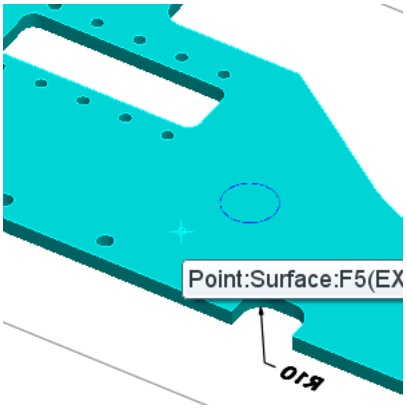
1. **Named View List** , **Datum_Target**, **Insert**, **Cosmetic**, **Designated Area**, select the sketch circle, **MMB** to confirm



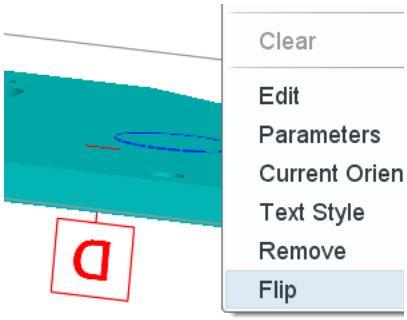
3. Select **Datum Target Annotation** , **OK** to Add Annotation, **Name**: D, check **Geometry** reference option, select the top surface of the part



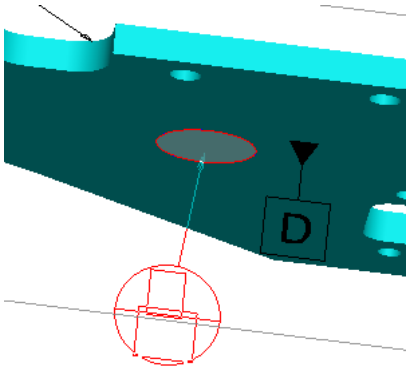
4. Select a point on the top surface to place the annotation



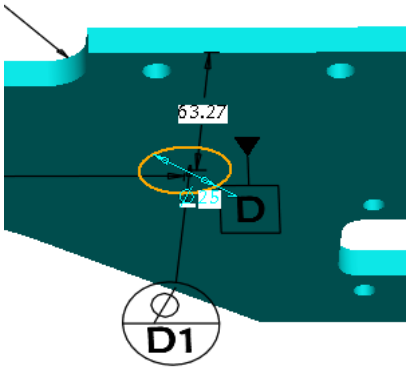
5. **OK** on Set Datum Tag dialog box, **RMB** on the graphics window, **Flip**



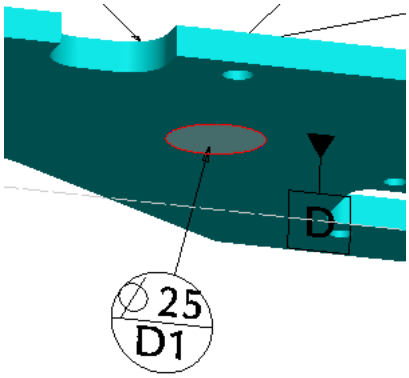
6. Select **Add** from the annotation definition dialog box, select **Browse…** to open defined symbol, double click **single**, select **circareatgt.sym**, **Open**, change **Next leader** type to **On Surface**, select the designated area, move the mouse away and **MMB** to confirm placement



7. Select **Variable Text** tab, **Pick Dimension**, select the radii 25 of the designated area, **OK**, **OK** to close Datum Target Annotation Feature dialog box



8. Select the datum target annotation, **RMB Select Reference**



9. **Window**, **Close**
10. **File, Erase, Not Displayed**

Pro/ENGINEER Manikin

Objective

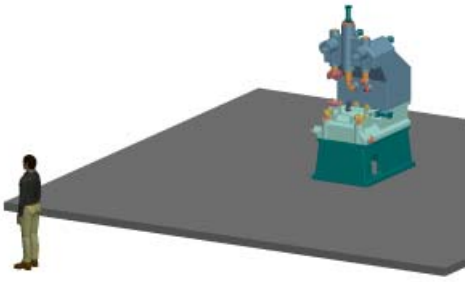
This tutorial will show how to access Pro/ENGINEER Manikin, place a manikin into a design assembly, position and define postures and run some simple human factors analysis.

Insert Manikin

The PTC Manikin library must be installed correctly to access the manikin specified below. This can be downloaded or ordered from the Technical Support Software Downloads page under Pro/ENGINEER. The ptc_maniking.asm included with this exercise has limitations.

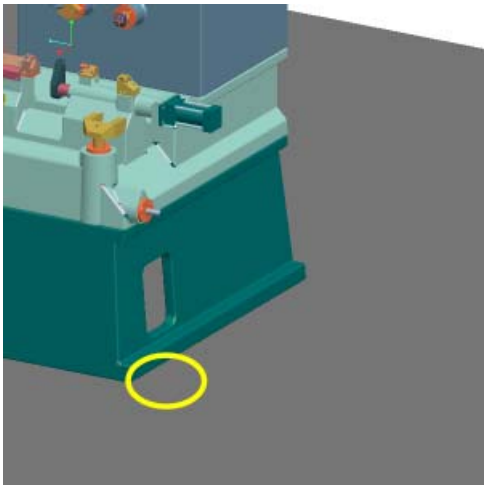
1. **File**, **Open**, **MANIKIN** folder, **working_zone.asm, Open**
2. Select **Insert, Manikin** to assemble a Manikin
3. Select **M_IT_50.ASM** from the population database and select **Open**.

The Manikin is added to your assembly; now move it near the workstation.



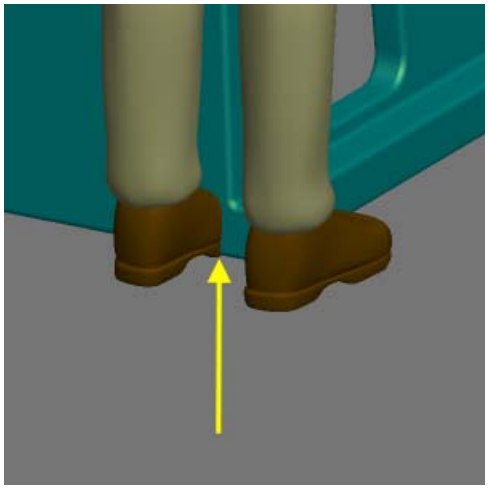
4. Saved View **View 2**
5. The Place Manikin dialog box defaults to a standing position and requires two references…
   - First, the right foot needs to be placed. Select the location by clicking on the floor as indicated by the yellow circle below.

- Next, you will select a plane that the Manikin will face. Select the surface indicated below.



6. Select **OK** in the **Place Manikin** dialog box
7. Saved View **View 3**

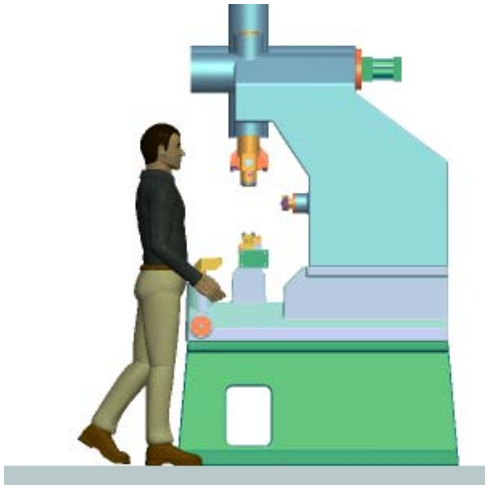
The left foot will interfere after initial placement; we will fix this in a moment. This interference allows the manikin to get close enough to the work cell equipment.



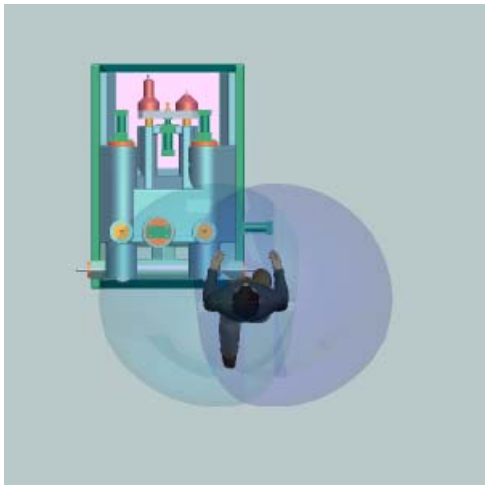
Posture and Reach

Now you need to apply an appropriate posture to the Manikin.

1. Select **Manikin > Apply Posture**
2. In the **Macro** folder, select the **CARRYING_BOX.MPD** posture and **Apply**

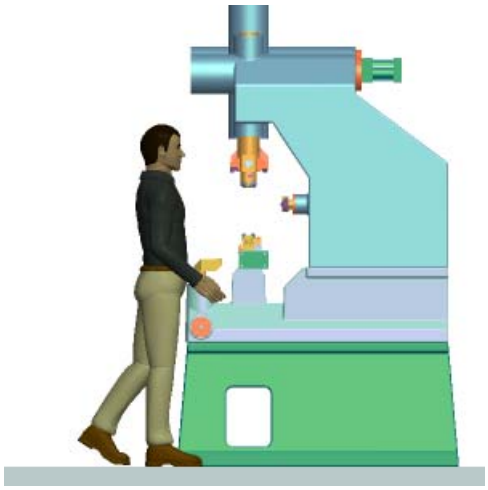


3. Toggle the Reach Envelope on; select **Manikin, Reach Envelope**
4. Saved View **View 4**

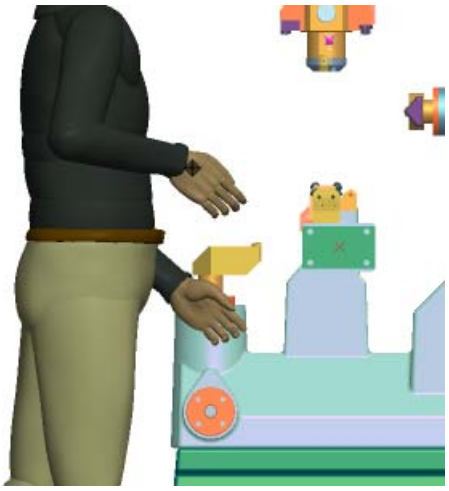


5. Toggle the reach envelopes off; select **Manikin, Reach Envelope**
6. Saved View **View 5**

In addition to applying postures to the Manikin, you can manipulate the Manikin into a desired position. You will use 2D drag to move the right hand and arm.



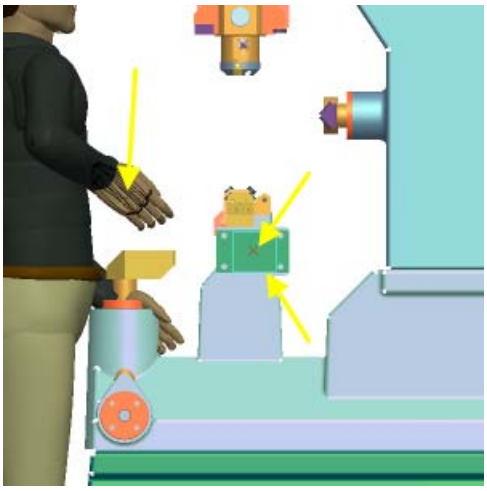
1. Select **Manikin, Manipulate**
2. In the Manikin Motion dialog box, select **2D Body Drag**. Click the middle of the right hand once and slowly move the mouse upward.



3. When the hand is in place click once, then select **Close** in the Manikin Motion dialog box.

Although this method is good for free-hand manipulation of your manikin, in this example the Reach tool will provide placement with greater precision.

4. Toggle **Point Display** on
5. Select **Manikin, Reach**
6. The Reach dialog box requires three references…
    - First, select the middle of the right hand
    - Next, select the point shown in the image below
    - Finally, select the end plane of the component that the Manikin is reaching for
7. Select **Close** in the Manikin Motion dialog box.

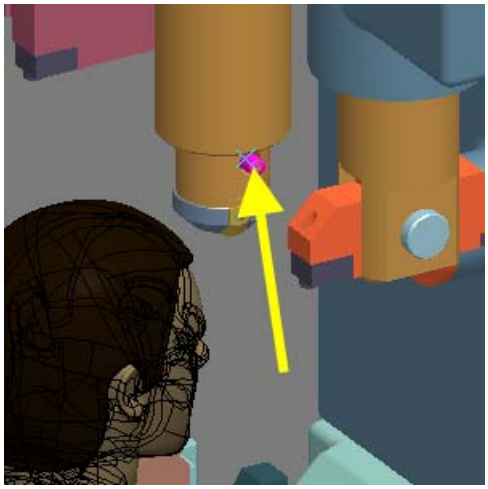


Vision

Now you will have the manikin look at a point in the assembly.

1. Saved View **View 6**
2. Select **Manikin, Look At**
3. The Look At dialog box requires two references…
    - Since your Manikin is the only one in this session it is automatically selected

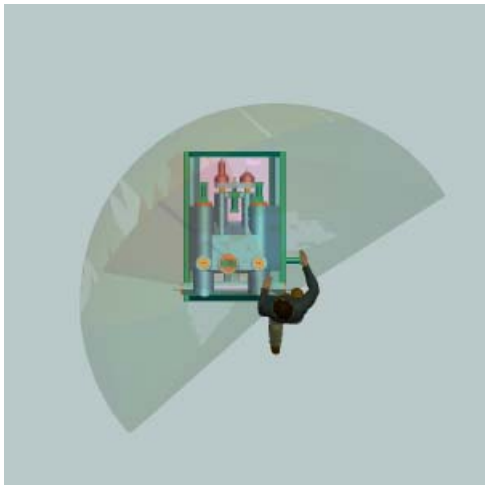
- Select the point circled in the image below



4. Select **Close** in the Manikin Motion dialog box
5. Toggle **Display Points** points off
6. Toggle the Vision Cones on; select **Manikin, Vision Cones**

> Vision cones for manikins are available at any time. They represent…
>
> - Peripheral vision (global vision)
> - Binocular (the visual field that can be seen by both eyes)
> - Optimal (operational zones)
> - Accurate (reading zone)

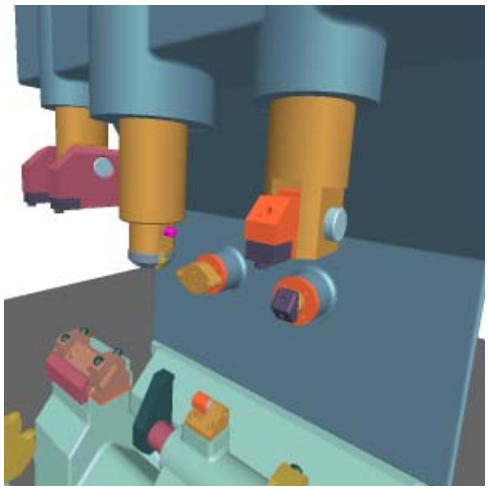
7. Saved View **View 4**



8. Toggle the vision cones off; **Manikin, Vision Cones**
9. Saved View **View 6**

> You can also view your assembly from the Manikin's viewpoint.

10. Toggle the Vision Window on; select **Manikin, Vision Window**

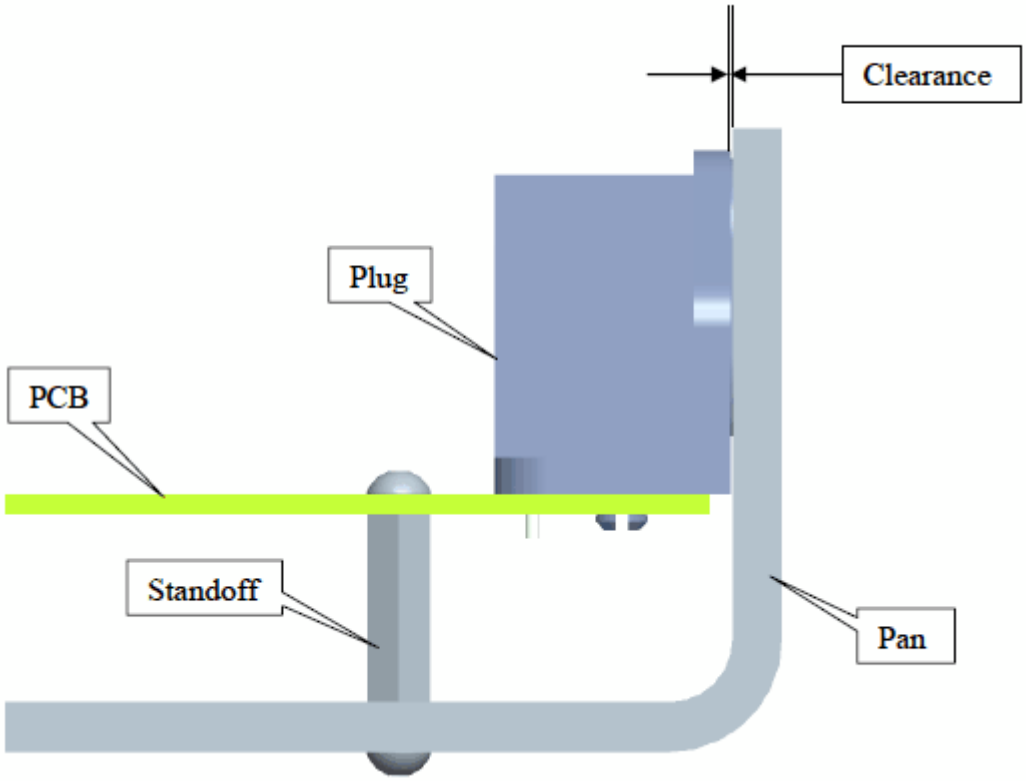


11. Toggle the vision window off; select **Manikin, Vision Window**
12. Saved View **View 7**
13. **Window, Close**
14. **File, Erase, Not Displayed**

Tolerance Analysis

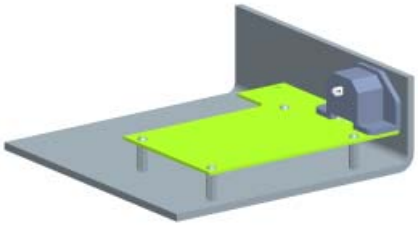
The objective of this analysis is to determine whether it is possible to assemble the PCB into the assembly. The current assembly process specifies that the bottom screws are to be installed and tightened first. Next, the plug is snapped into holes in the PCB

Then the PCB assembly is placed into the assembly and the top screws inserted and tightened. There should be a minimal gap between the plug and the pan, but an interference condition could prevent the ability to insert the screws into the PCB



Open Model and Set the View

1. **File**, **Open**, **TOLERANCE-ANALYSIS** folder, **circuit-card.asm**, **Open**
2. Saved View **ISO**

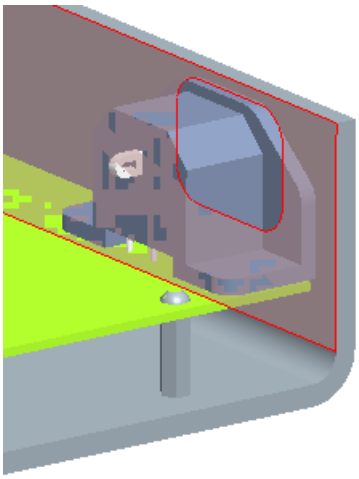


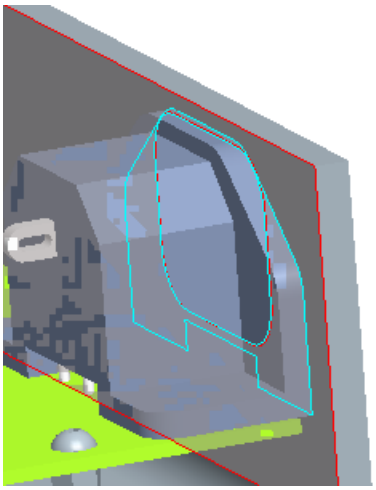
Initiate Tolerance Analysis Measurement

1. **Analysis**, **Tolerance Study**

> The Tolerance Analysis Manager dialog box lists all of the existing tolerance analysis measurements in this model. You can add, edit, and delete tolerance analysis measurements from this dialog box.

2. Select the Add icon in the Tolerance Analysis Manager dialog box.
3. Now you will need to select two entities…
   - First, select the inside vertical surface of the pan **Surf:F1(FIRST FEATURE):PAN** (see the figure below).
   - Select the hidden vertical surface of the plug **Surf:F4(PROTUSION):REC-3PIN** (see the figure below). Use Query Select or Pick from List.



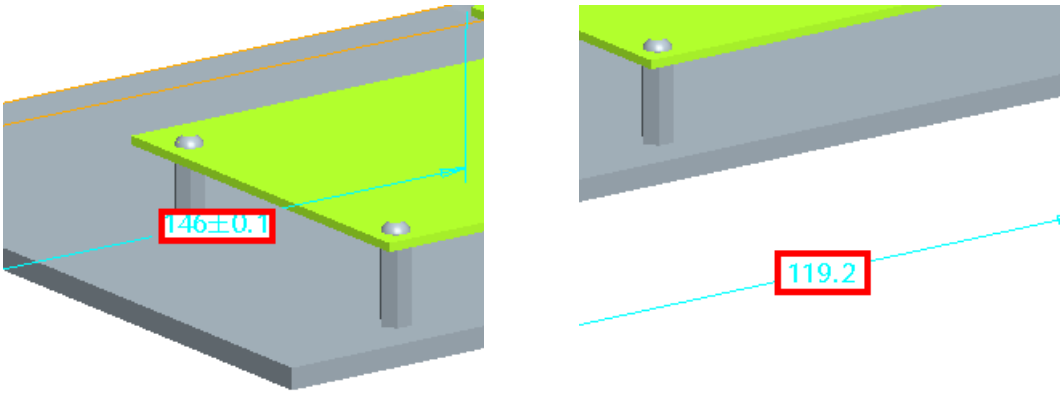


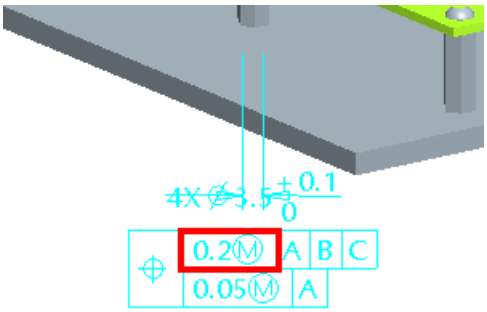
Restarting a Measurement
If you make a mistake and you want to start over, you can right-click in the Measurement Table view and select **Restart Measure** from the shortcut menu

Select Dimensions

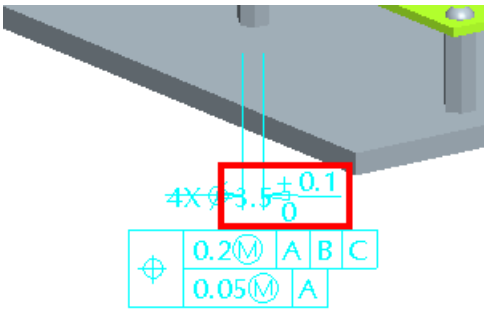
Redisplaying the Candidate Dimensions
During the dimension selection process, the candidate dimensions for the active part are automatically displayed. You can change active parts by clicking a new part. If you don't see the expected dimensions, the wrong part may be the active part. In that case, click the part from which you need to select dimensions

1. Now you need to select four dimensions…
   A. First, select **146 +/- 0.1**, the pan length
   B. Next, select **119.2**, the basic locating dimension for the screw hole
   C. Then, select **0.2**, the position tolerance for the hole pattern
   D. Last, select **3.5 +.1/-0**, the hole diameter





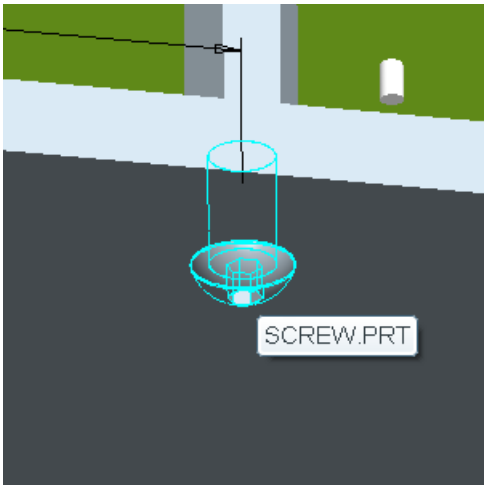
C

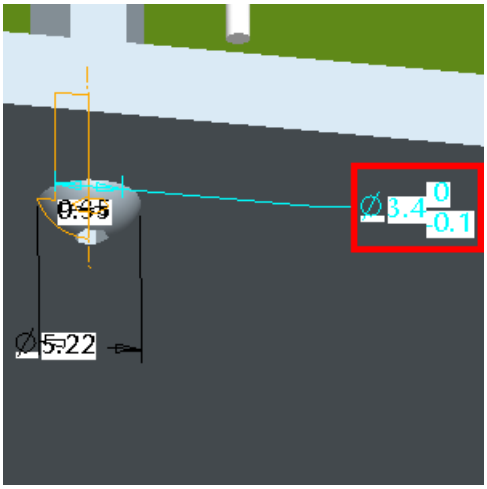


D

If you accidentally select the wrong dimension, you don't have to start over. During dimension selection, select Cancel in the Select menu. You have a number of cancel options (shown in the Pro/ENGINEER prompt area). Option 1 will undo the last selection and resume the selection process.

2. In the Select Option dialog box, select **Selected Cylinder is a Hole**, then hit **OK**
3. Saved View **UNDERNEATH**
4. Click the bottom screw; then select **3.4 +0/-.1**, the major diameter of the threads

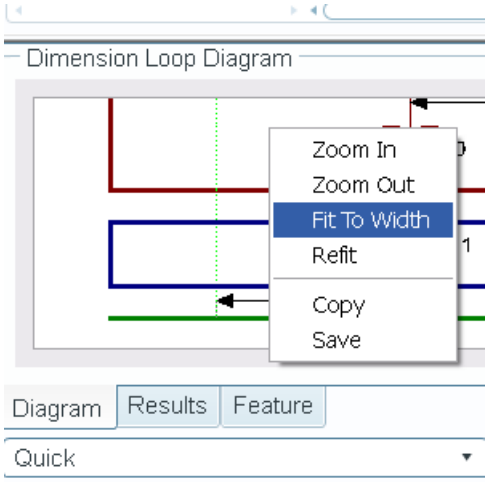




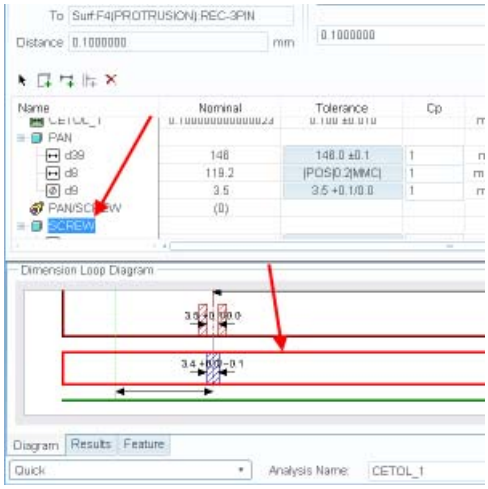
5. In the Select Option dialog box, select **Selected Cylinder is a Pin**, then **OK**

Explore the Tolerance Analysis Extension

1. **RMB Fit to Width** in the Dimension Loop Diagram

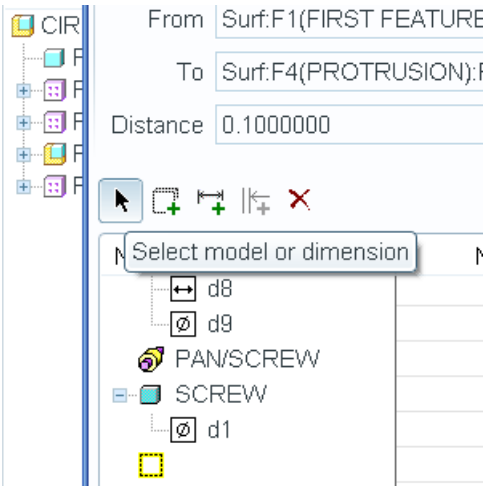


2. Click the various objects in the Name column of Measurement Table. Then click the various objects in the Dimension Loop Diagram. Notice the cross-highlighting between the two

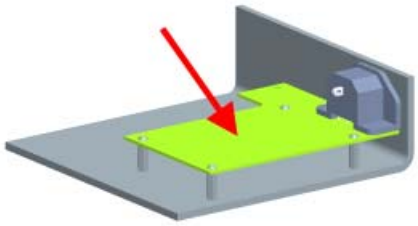


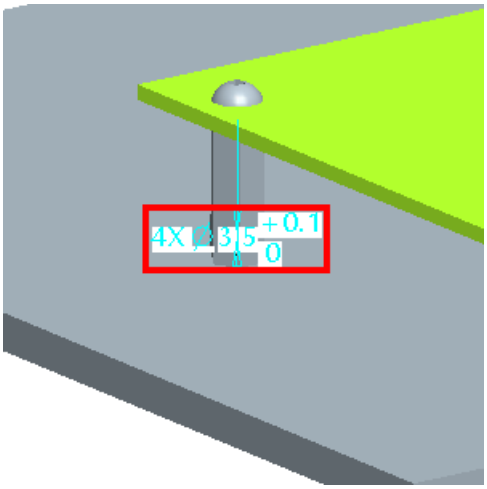
Resume the Dimension Selection Process

1. Saved View **ISO**
2. Choose the Select Model or Dimension icon in the Tolerance Analysis dialog box

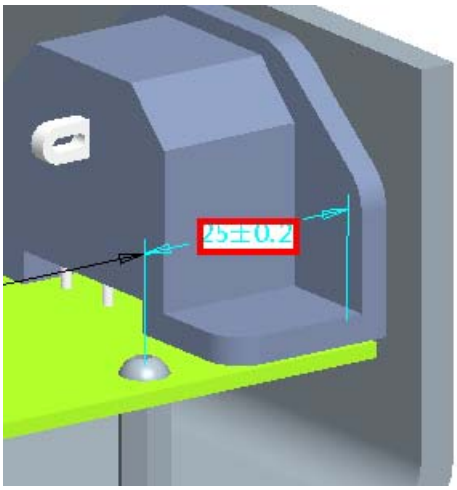


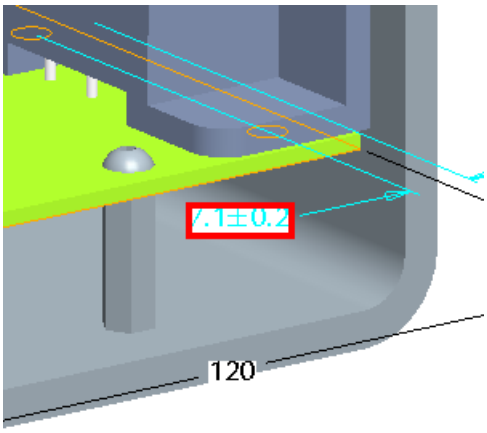
3. Click the **PCB** to display the candidate dimensions for that part; then select **3.5 +0.1/-0**, the hole diameter



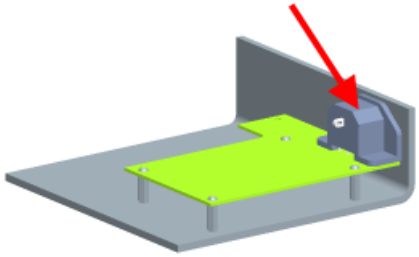


4. In the Select Option dialog box, select **Selected Cylinder is a Hole**, then **OK**
5. Now you need to select two dimensions…
    - First, select 25 +/- 0.2, the screw hole location
    - Next, select 7.1 +/- 0.2, the plug hole location

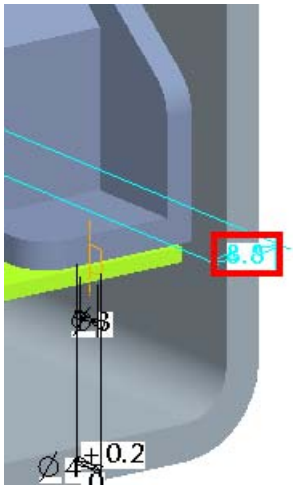


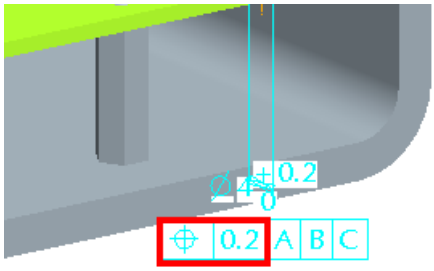


6. Click the Plug to display the candidate dimensions for that part



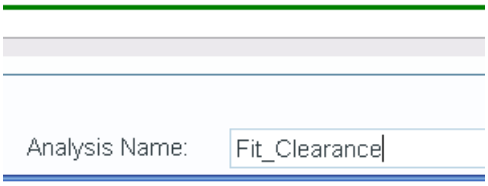
7. Now you need to select two dimensions…
   - First, select **8.8**, the basic locating dimension for the snap fit
   - Next, select **0.2**, the position tolerance for the snap fit



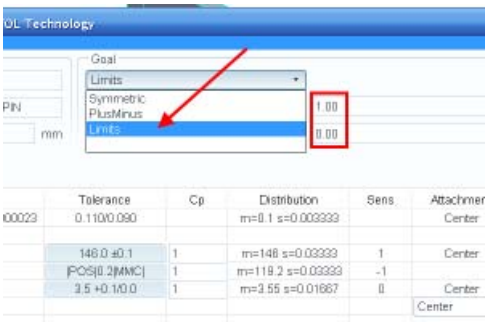


Specify Measurement Limits

1. Click **Cancel** in Select dialog box to pause the selection process
2. Rename the analysis to **Fit_Clearance** by typing in the text field at the bottom of the Tolerance Analysis dialog box



3. Under the Goal heading in the Tolerance Analysis dialog box select Limit. Enter an upper value of **1.0** and a lower value of **0.0**

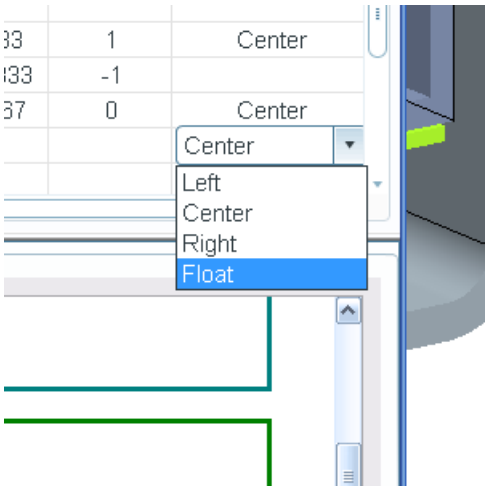


Modify the Interface Properties

Whenever the last feature of one part and the first feature on the following part are a pin and hole, the application creates a pin/hole interface. By default, the pin is assumed to be centered in the hole. However, the clearance between the pin and the hole is often an important consideration in a tolerance analysis, so you have a number of options for how to represent that interface: centered right (tangent), left (tangent), or floating

First, consider the screw in the pan. The screw is inserted into the hole and tightened down in the standoff, so the screw is randomly located within the clearance of the hole. This situation is best represented with the float option

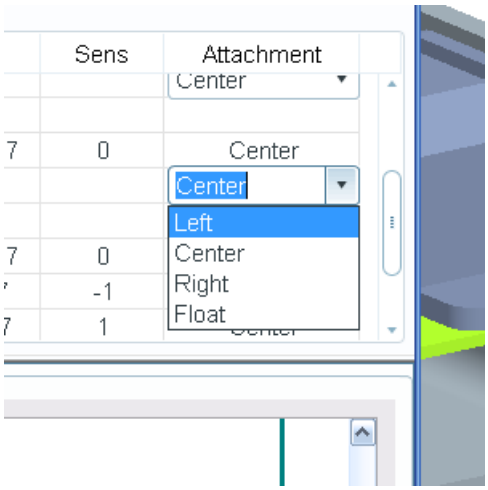
1. Select **Float** from the Attachment column for Pan/Screw below Pan



- With the **Float** command, the application introduces a variable that represents the random variation of the location of the pin in the hole. Notice that a double-arrow in the Dimension Loop Diagram indicates that this interface is floating

Now, consider the screw in the PCB. In this case, the screws are inserted into the holes and loosely threaded into the standoffs. Before the screws are tightened, the PCB can slide left or right until a hole comes in contact with a screw. Since we are trying to predict the probability of fit for this case, we should set the properties of this pin/hole interface to maximize the clearance between the plug and the pan.

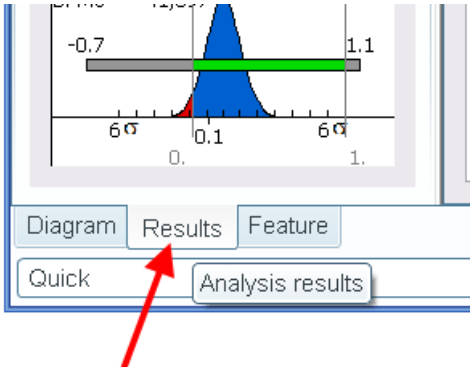
2. Select **Left** from the Attachment column for Screw/PCB below Screw



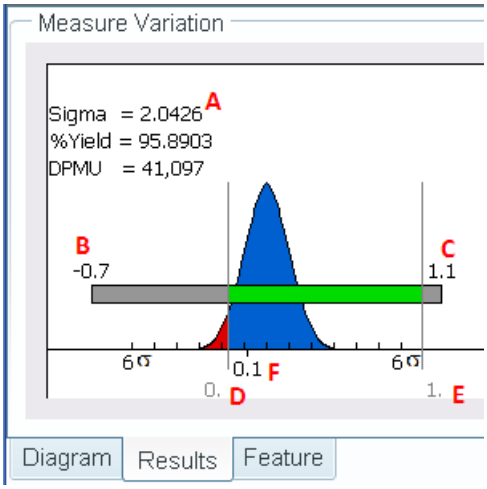
Notice that the vertical line joining the pin and hole in the dimension loop diagram is at the left tangent location. The PCB is pushed away from the interface that we are measuring (indicated by the dashed vertical lines) so that the PCB hole is touching the pin on the left side. Notice also that the nominal value of the measurement changes when you change the interface from centered to left tangent.

Examine the Results

1. Click the **Results** tab at the bottom of Tolerance Analysis dialog box to view the analysis results

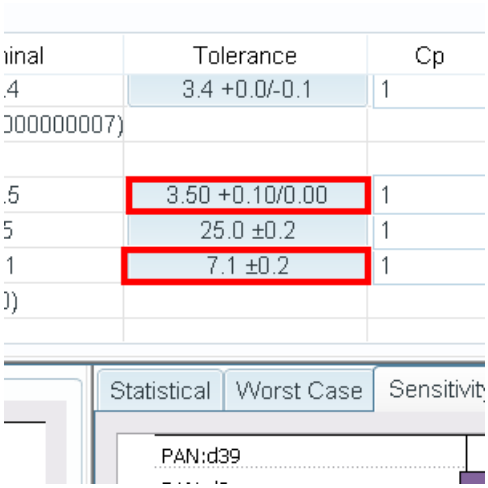


The **Analysis Results** view has two display regions. The left region shows a variation plot for the measurement. The variation plot shows the statistical variation plot and the worst-case range for the measurement based on the specified tolerances for the dimensions in the dimension loop.

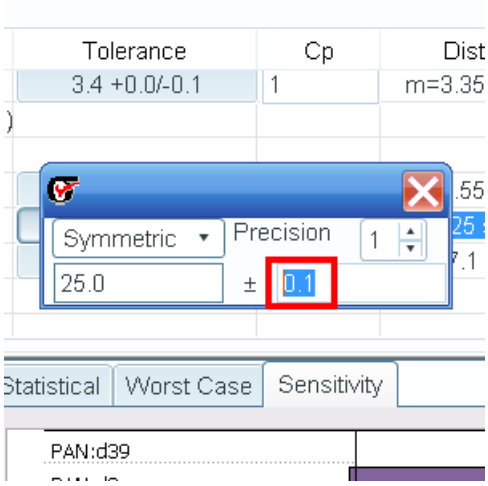


A. Mean and Standard Deviation
B. Min. WC value of Measurement
C. Max. WC Value of Measurement
D. Lower Measurement Limit
E. Upper Measurement Limit
F. Measurement Target Value

The right region of the **Analysis Results** view is a tabbed display of contribution and sensitivity plots. For example, the Statistical Contribution plot shows the percent contribution of each dimension to the variance of the measurement. The results indicate that two of the dimensions from the PCB are the largest contributors (d266 and d43).

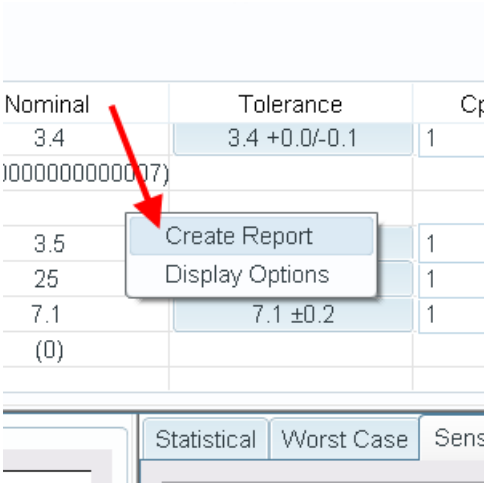


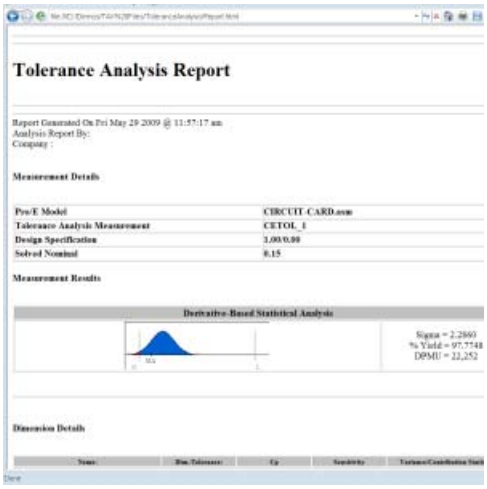
2. Click the tolerance field for the *d266* dimension in the PCB. Change the value to **0.1** and press the **Enter** key. Then **Close** the input dialog box. Repeat these steps for the *d43* dimension in the PCB



Notice that when you make a change to the tolerance, the results automatically update. You can continue to change the tolerance values until you get the desired measurement variation. Note that the actual Pro/ENGINEER dimension properties are not modified until you close the Tolerance Analysis Extension Powered by CETOL Technology interface and accept the changes.

3. **RMB Create Report** in the Measurement Table, ✔ and **Close**





The report is displayed in the Pro/ENGINEER browser. Click in the Tolerance Analysis Extension Powered by CETOL Technology window to save the tolerance analysis measurement feature, update the modified Pro/ENGINEER tolerances and close the application

4. **Window**, **Close**
5. **File**, **Erase**, **Not Displayed**

Congratulations! You have completed the tutorials in this Hands On Workshop! Thank you for your participation and we look forward to having you attend another Hands On Workshop in the future.