

STEPS TO BUILDING A SHEET METAL CHASSIS FOR YOUR 2.810 CAR USING SOLIDWORKS

By Eddy Reif, Laboratory for Manufacturing and Productivity, MIT August 1998

In the next pages you will find the basic steps to create your own car chassis using the SolidWorks CAD software. The basic idea is to first sketch out your solid in two dimensions, and then create the third dimension by extruding. Design your chassis as a 3D object made of a thin sheet. Solidworks will unbend it for you to create the pattern to be cut on the waterjet. Bending up of the part will be done on the pan press and brake press in building 35.

STEP 1.- Creating the base of the chassis

In this step you'll create the main element of the chassis, upon which you shall later insert all your features. Since this is the very first step, we'll go through it with detailed explanations.

1-A) Creating the base with folds that have a bend line that crosses the part sideways.

1.- Open the View Orientation window by clicking on **View and Orientation**.

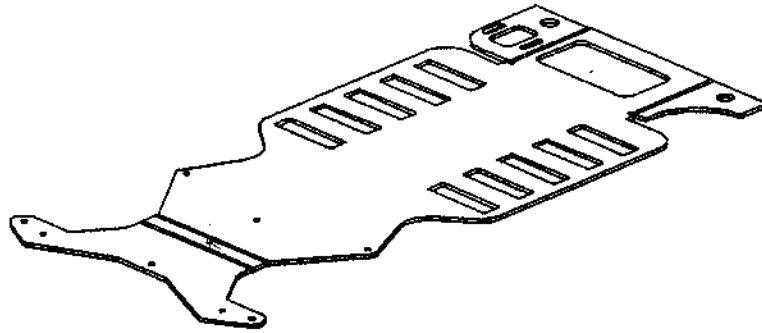
Tip: Click the icon at the upper left corner of the view orientation window so it will always remain where you place it.

2.- Select the **Right** view, by double clicking on it on the View Orientation window.

3.- Select **Plane 3** in the Feature Manager Design Tree (*from now on referred to as the FMDT*).

4.- Click the sketch button on your toolbars. (It is the one with the pencil on it).

- 4) To flatten click the rollback bar at the bottom of the FMDT, the cursor shape changes to a hand. Drag the bar up until it is above Process-Bends1.



Step 4.- Saving your unbent part as a Drawing file (.drw. .slddrw)

You will probably want to export your unbent (flat) drawing to MasterCam for machining or to the waterjet. The best way to do this is as a .dxf file. But you can't create a .dxf file directly from a part in SolidWorks it has to be from a drawing. You can do this by suppressing the Process-Bends1 step.

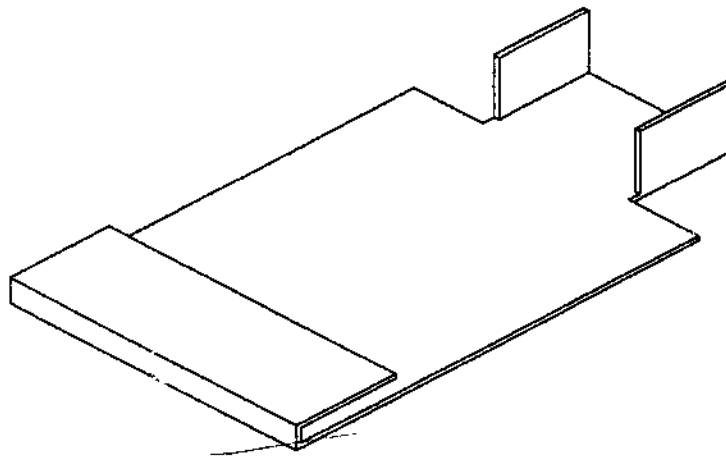
- 1) Select in the FMDT the Process-Bends1 feature.
- 2) Click the suppress button on the Dependency Editing toolbar.
- 3) To unsuppress click the unsuppress button in the same toolbar.
- 4) Open a drawing file by clicking **File-New** and selecting **Drawing**.
- 5) The Template To Use window appears. Depending on the size of your chassis you will have to change the default A-landscape paper size to another one, probably B-Landscape. And you can choose whether you want the standard template already in your drawing, or you can start with no template. After adjusting your sheet click OK.
- 6) If you decided to insert the Standard Template, you may edit it by clicking on **Edit-Template** on the main menu.
- 7) Click **Edit-Sheet** to prepare your sheet to insert the part.

2.- Sketch the boss as you would see it on your design by looking at it from above. Remember that one of its sides should be coincident with one of the base's edge because that is going to be the folding line, and that the dimension that is perpendicular to the base edge should have the same value as the thickness of the sheet.

3.- Click **Inert-Boss-Extrude**. The Extrude Feature window appears so that the type and depth can be selected. In Type choose blind, and the depth should be the height of the boss that is about to be extruded.

4.- Click OK.

By doing steps 1-A) and 1-B) as necessary according to the design, the base of the chassis is now created, and is ready to now have all the features inserted in to it.

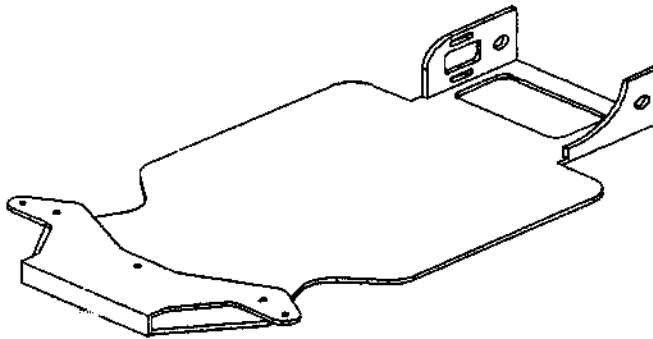


At this phase we have cut (explained below) and extruded back mounts for the motor
Step 2.- Insert features

This is where the designing really starts, in this step the procedures for the three basic features will be explained. They are:

- Cutting.

- 1) Select the edge (click on it), face or loop to be filleted.
- 2) Click on the fillet button in the features toolbar, the Fillet Feature window appears.
- 3) Choose the desired radius.
- 4) Verify that the number of edges, faces or loops that need to be filleted are listed in the Items To Fillet box.
- 5) Finally select the fillet type, whether its Constant Radius, Variable Radius or face blend and click OK.



The small fillets are hard to see in this figure

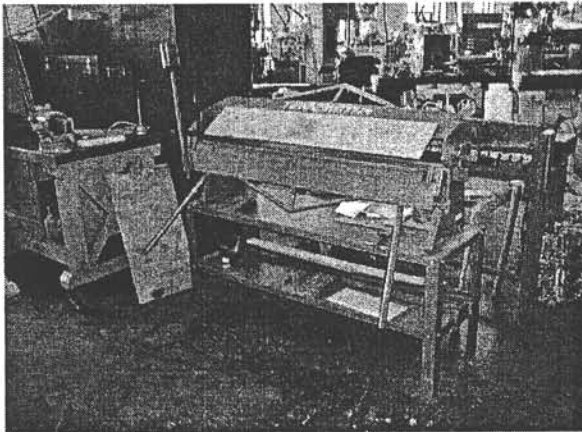
Creating Patterns:

This can be a very useful feature, and if used correctly can save the trouble of making the same feature many times, when a pattern is part of the design.

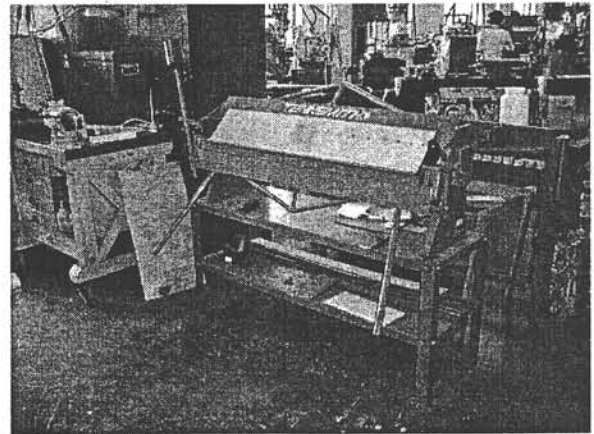
- 1) First create a single feature, for example a cut or a boss, that will repeat itself in a pattern, it can be in a circular or in a linear pattern.
- 2) Select the feature just created, whether its in the FMDT or in the drawing.
- 3) Click **Insert-Pattern/Mirror-Linear (or Circular) Pattern**, the Linear (or Circular) Pattern dialog box appears.
- 4) In any case choose the number of instances to be made.

2.810 Fundamentals of Manufacturing Processes

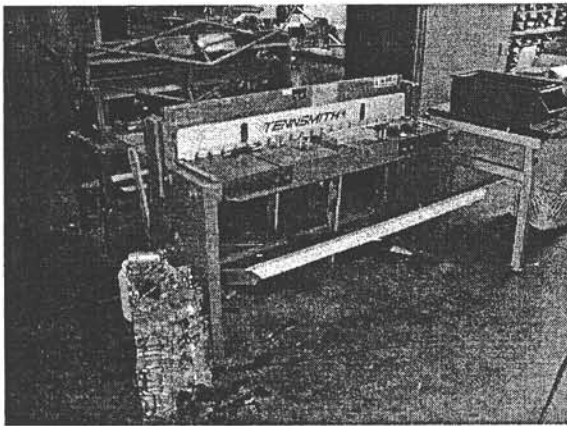
Sheetmetal Working Machinery



Fingers Brake (open)



Fingers Brake (closed)

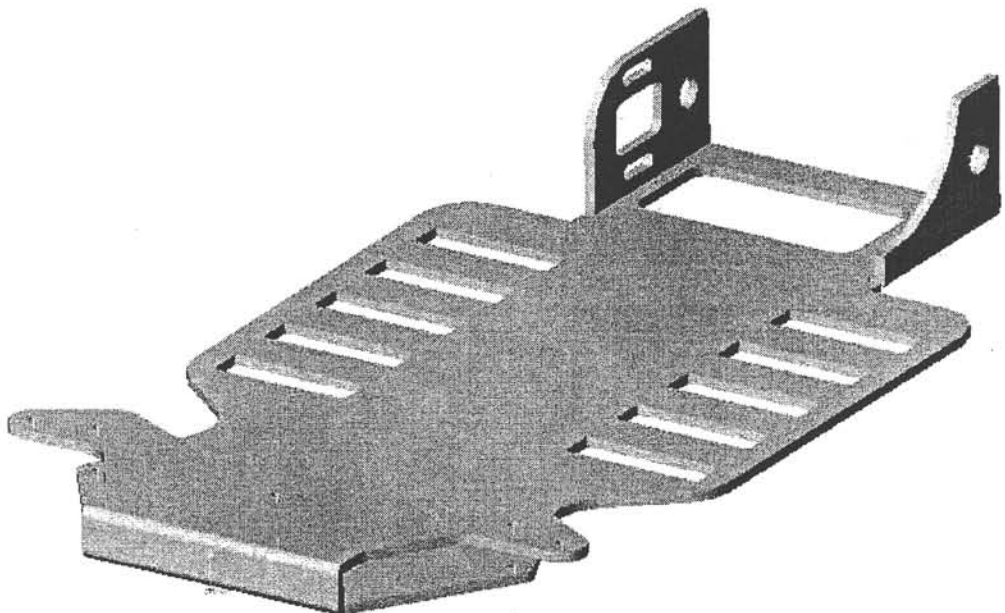


Foot Shear



Notcher and Rod Bender

- base of chassis (Unbent Bar)
- Front
- Top
- Side
- Origin
- Base-Extrude-Thin
- Cut-Extrude1
- Cut-Extrude2
- Cut-Extrude3
- Cut-Extrude4
- Cut-Extrude6
- Cut-Extrude13
- Cut-Extrude7
- Cut-Extrude8
- Cut-Extrude9
- Screw-holes1
- Back-Cut1
- Motor-Holders
- Rear-Cut1
- Rear-Fillet1
- Rear-Cut2
- Rear-Cut3
- Fillet1
- Fillet2
- Cut-Extrude5
- LPattern1
- LPattern2
- Fillet3
- Fillet4
- Axis-Hole
- Rear-Fillet2
- Rear-Fillet3
- Screw-holes2
- Fillet6



View Orientation

<input checked="" type="checkbox"/> *Normal To	Add...
<input checked="" type="checkbox"/> *Front	Update
<input checked="" type="checkbox"/> *Back	Reset
<input checked="" type="checkbox"/> *Left	Help
<input checked="" type="checkbox"/> *Right	
<input checked="" type="checkbox"/> *Top	
<input checked="" type="checkbox"/> *Bottom	
<input checked="" type="checkbox"/> *Isometric	
<input checked="" type="checkbox"/> *Trimetric	
<input checked="" type="checkbox"/> *Dimetric	

- 8) Click on the Named View icon on the Drawing toolbar.
- 9) On the main menu select **Window-*the name of the file with the part.***
- 10) Your part file appears, click anywhere on your part. The Drawing View-Named View dialog box appears.
- 11) Double-click on any of the names of the list, or on Current for the view that is on the screen now.
- 12) On the Window menu, click on the name of your drawing so you can bring the drawing document forward.
- 13) Click in the drawing where you want to place the part view.
- 14) Click on **File-Save As** on the main menu.
- 15) The Save As window appears, where you can select the .dxf type of file.

