



Autodesk  
University  
2007

# Sheet Metal: Autodesk® Inventor™ on the Cutting Edge

Walt Jaquith – Simplex Manufacturing

**MA315-3** Autodesk didn't just make some improvements to Autodesk Inventor's sheet metal application -- they took it to a whole new level. While the enhancements to the tools themselves are significant, even more important are the concepts behind the changes, which redefine Inventor's role in the design and manufacturing process. If you want to get a grasp on the new tools -- from the grand concept to the nuts-and-bolts functionality -- this class is for you.

## About the Speaker:

Walt is an independent design contractor in the aerospace industry. A user of Autodesk Inventor since its beginning, he has also been using Autodesk 3D design products for 14 years. Prior to his work in the aerospace industry, Walt worked "the other side of the wall" as a fabricator, mechanic, and general troubleshooter. He has experience in most mechanical engineering disciplines, and has served as a manager of CAD departments and fabrication crews. [wjaquith@comcast.net](mailto:wjaquith@comcast.net)



Autodesk  
University  
2007

## Sheet Metal: Autodesk® Inventor™ on the Cutting Edge

### COURSE INTRODUCTION

Release 2008 of Autodesk Inventor introduced sweeping changes in Inventor's sheet metal module. These enhancements reflect a fundamental shift in how Inventor not only approaches the design, but also supports the manufacture of sheet metal components. The list of additions and changes is quite long, and spans every aspect of sheet metal design. The clear goal is to position Inventor as a top contender in the sheet metal design world.

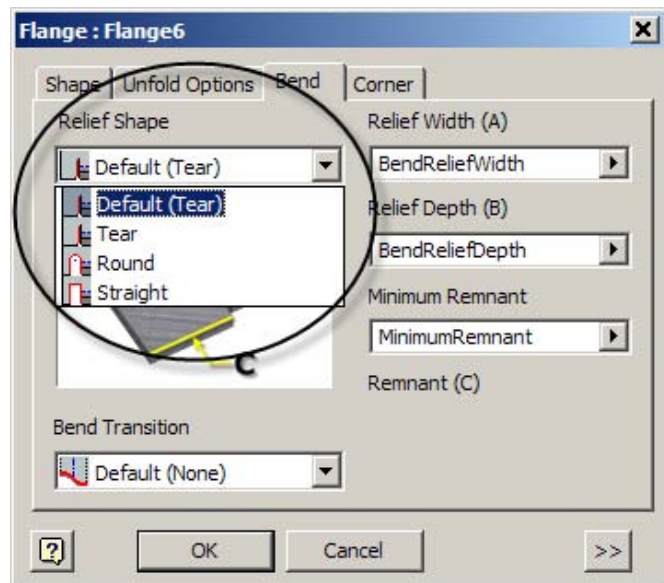
Accompanying the overhaul of the sheet metal module is a comprehensive set of help files; easily the best help documentation yet to be found in Inventor. This course does not intend to duplicate the excellent work that was done there. Rather it is intended to give a conceptual view of the new features and enhancements. The goal is to gain an understanding of the intent behind the additions, so that they can be used to their fullest potential in the design process.

### A QUICK LOOK AT SHEET METAL STYLES

As with most Inventor functions, the settings in the Style Manager play a major role in determining how Inventor's sheet metal tools function. The functionality of the Sheet Metal Style Editor is straightforward, and doesn't require a great deal of explanation beyond that which is offered in the help files. However, there are a few things worth noting here.

#### **Default Option Behavior**

Default options for each of the choices in the various drop-down menus can be selected in the styles editor. Once a particular choice has been made, that will be the tool's default option. If the menu is pulled down, the user will notice that the chosen option is listed in the menu twice; once by itself, and once as the default option from the style (FIG 1). If the latter is selected, the tool will default to the choice set in the active style, and if the style is changed that feature will



**FIG 1: ON THE BEND OPTIONS TAB, THE RELIEF SHAPE PULL-DOWN SHOWS THE DEFAULT OPTION FROM THE STYLE (WHICH IS "TEAR"), AND ALSO OFFERS TEAR AS AN OPTION THAT IS NOT LINKED TO THE STYLE.**



change with it. If the former is selected, the option is the style is overridden, even though it is identical to the default. In that case, if the style's setting is changed to some other option, the selection for that feature will not change.

## Sheet Metal Layers

Several new layers have been added to accommodate some of the new sheet metal documentation objects. These layers include up and down sheet metal bends, bend extents, and sheet metal punch

centers (FIG 2). Up and down bend lines are on separate layers so that they can be assigned different colors, adding one more method of enhancing clarity in the drawing.

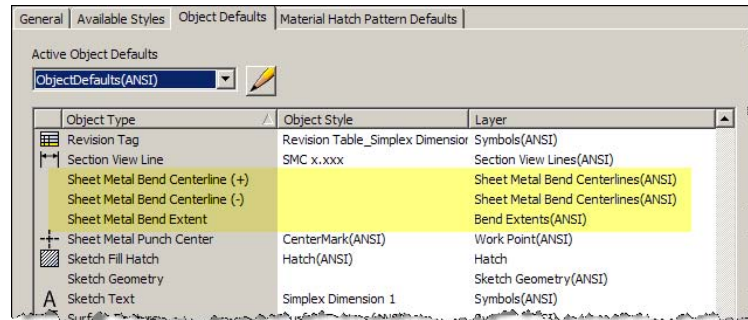


FIG 2: SHEET METAL LAYER OBJECTS SHOWN UNDER OBJECT DEFAULTS.

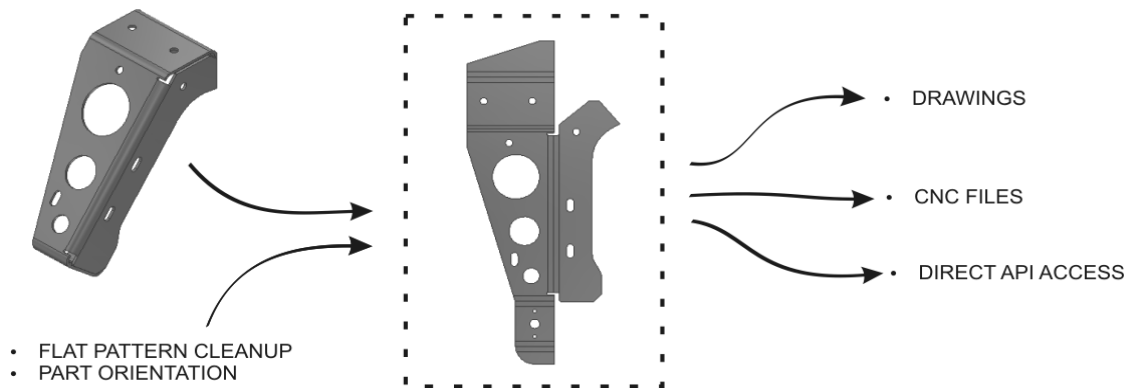
## THE FLAT PATTERN ENVIRONMENT

The flat pattern environment within Inventor has received perhaps the most significant yet the least obvious changes of all the sheet metal features. A totally new “part within a part” environment was developed for sheet metal flat patterns. In addition to being a functional modeling environment in itself, the new flat pattern was designed to be a data container for communication with downstream manufacturing processes. This important concept lies at the root of all the work that has been done in Inventor’s sheet metal module.

### The Flat Pattern as a Container for Manufacturing Data

Modern sheet metal manufacturing incorporates several unique requirements in dealing with manufacturing data. The process often uses automated machinery with CNC (Computer Numerical Control) to cut, punch, and form sheet metal components. These machines read files that can be generated by CADD packages such as Inventor. In the past, the process of generating readable files for sheet metal manufacturing machinery has been a hit-and-miss proposition. One common problem has been the generation of incompatible file types. In such cases the CNC file would be unreadable by the target machine. Another problem was the formatting of the data within the file. CNC cutting and forming machines typically require that data be presented to them in a very particular manner. The output capability of the design software has often been too generic, resulting in files that required a great deal of post-process cleanup and ‘tweaking’ to make them viable. Often these secondary operations required a different software package such as Autodesk AutoCAD.

File and data compatibility issues are not the only cause of the need for CNC file cleanup. Limitations inherent to the 3D solid modeling process sometimes create the need to add or subtract features from the flat pattern of the part. This occurs mainly



**FIG 3: THE FLAT PATTERN ENVIRONMENT NOW HOLDS A MUCH MORE IMPORTANT PLACE IN THE SHEET METAL MODULE. IT FORMS A “CONTAINER” WHICH IS DESIGNED TO HOLD ALL NECESSARY INFORMATION TO DETAIL, CUT AND FORM THE PART**

because the manufacturing process sometimes allows limited warping of the sheet material to obtain the desired shape. The production of a tight corner seam is a common example of this. In order for the joint to be tightly closed when formed, the metal at the corner must be allowed to warp in a manner beyond the normal stretching that occurs in a standard two-dimensional bend. This localized warping normally poses no problem for the manufacture of a sheet metal part. However, it does present a larger problem for 3D design software. Calculating the random shapes that a formed sheet metal part might take during a warping process would place an extraordinary load on the solid modeling engine. This computational overhead would be incurred for no good purpose, because the shape of the material at the warp cannot be accurately predicted or controlled. Therefore, the justification for adding that overhead comes down to modeling aesthetics. If the solid modeler could accurately process the warping of the sheet metal material, the folded model of the part in the viewing window would appear a bit more accurate. In the end, this marginal improvement in visual fidelity is simply not worth the cost in performance that it would incur.

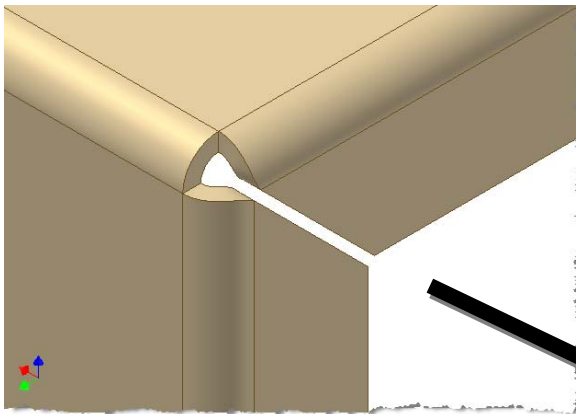
### ***Flat Pattern Cleanup***

The solution to this problem comes in the form of flat pattern cleanup. The folded model is created in ways that avoid the issue of warping and its associated overhead. This results in a slightly inaccurate flat pattern when the part is unfolded. Therefore, it becomes necessary to modify the flat pattern to make up for the solid modeler's limited ability to model warped material. As is the case with the cleanup necessitated by file compatibility issues, these post-processing operations have often been conducted in AutoCAD rather than in Inventor. This need for a second application is in direct conflict

with Autodesk's goal of developing the Inventor flat pattern environment as a complete container for sheet metal manufacturing. In developing solutions to the issue of flat pattern cleanup, Inventor 2008 has taken a two-pronged approach.

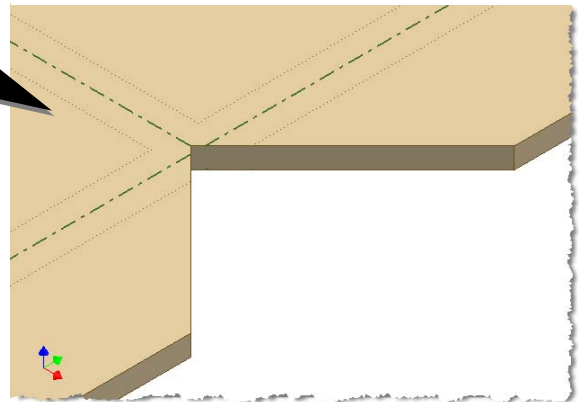
### ***Automatic Flat Pattern Correction***

The first part of the solution to the flat pattern cleanup problem comes in the form of automating the cleanup for the most common sources of flat pattern inaccuracies. As mentioned above, tight corner seams are a primary culprit. Bend transitions also have this tendency to warp material at the corner, and in both cases the issue has been



addressed in the same way. Inventor now offers four types of corner seam conditions, and five types of bend transitions, most of which cause some 3D warping in the material, and so cannot be accurately and easily represented in the folded model. In these cases the corner seams are represented in

FIG 4: THIS JACOBI CORNER CONDITION IS SET TO "INTERSECTION". TO AVOID MODELING WARPED MATERIAL, THE FOLDED MODEL REPRESENTS THE CORNER AS "NO REPLACEMENT" HOWEVER, THE FLAT PATTERN SHOWS THE CORRECT CORNER SHAPE.



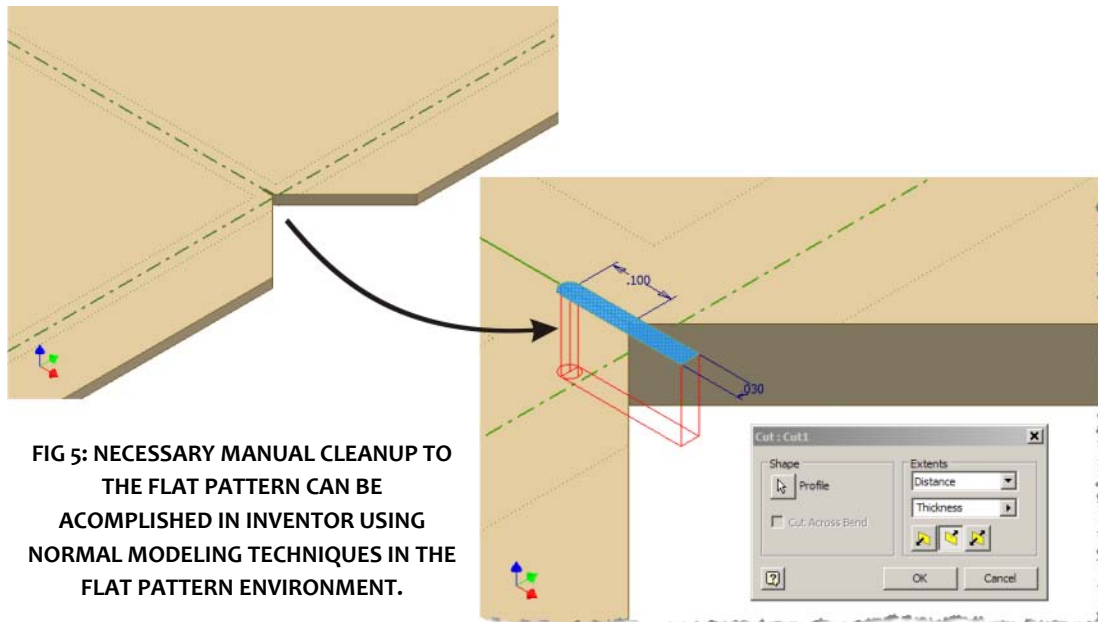
the folded model as "No Replacement".

For a bend transition, the folded model displays the transition as "none". These options simply remove all the material from the potential warping zone, and so impose no unusual demands on the computer. However, each corner seam and bend transition type will always be accurately represented in the flat pattern. Inventor automatically adjusts the flat pattern model as needed. In this manner, the flat pattern geometry is always presented correctly to downstream manufacturing.

### ***Manual Flat Pattern Correction***

The second part of the solution brings the remainder of the cleanup process into Inventor where it can be completed in a familiar and manageable environment. No matter how good the design software, there will always be cases where some manual cleanup of the flat pattern is necessary. Autodesk has addressed the issue by adding modeling capabilities to Inventor's flat pattern environment.

Inventor's sheet metal flat pattern is now itself a 3D model. It has mass properties and a thickness set by the current sheet metal style. Any needed cleanup to the flat pattern can now be accomplished using familiar Inventor part features. The intent of this new capability is not to make radical changes to the flat pattern. Since changes in the flat pattern are not reflected in the folded model, this would cause the folded model to be an inaccurate representation of the finished part. The intent is to allow the typically minor changes that represent areas where small amounts of warpage can occur, making them difficult or impractical to model in the folded version of the part.



**FIG 5: NECESSARY MANUAL CLEANUP TO THE FLAT PATTERN CAN BE ACCOMPLISHED IN INVENTOR USING NORMAL MODELING TECHNIQUES IN THE FLAT PATTERN ENVIRONMENT.**

The goal of the new internalized flat pattern cleanup functionality is to create a flat pattern within Inventor that is exactly what is needed to manufacture the part. In this way, a secondary application is no longer necessary, and the data available within Inventor itself is reliable. This has significant implications, as the data within the flat pattern environment has been exposed to the API. That means that it's possible to access flat pattern manufacturing information programmatically. This has the potential to eliminate the need to export files for consumption by CNC machines.

## **SPECIAL SHEET METAL FEATURES**

Sheet metal presents several totally unique requirements to the design process, some of which (such as the need to unfold a part) we've covered as major features. However, there are other equally important elements of sheet metal design that must be dealt with in whichever form of documentation is chosen. Several new or improved features in R2008 deal directly with these issues.



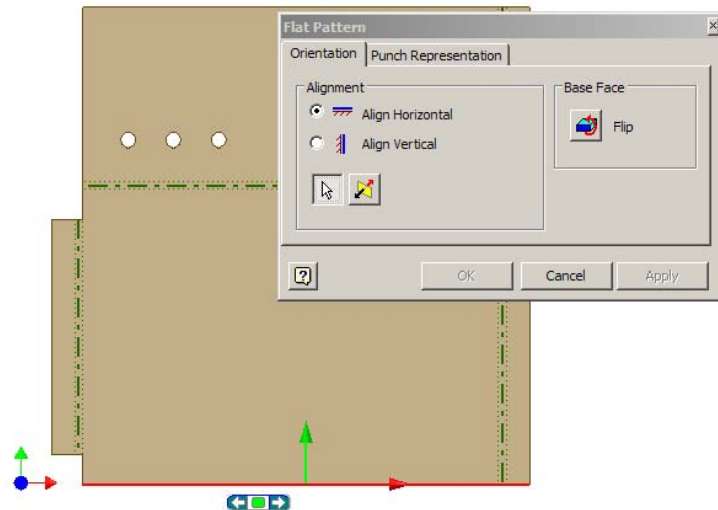
## ***Sheet Metal Grain Direction***

In a sheet metal component, the relationship between the various features and the extruded direction (sometimes referred to as the “grain”) of the material is often very important. Therefore, it is desirable to establish a coordinate system for the flat pattern that is totally independent of the folded model. This coordinate system has several uses, one of them being to orient the cutting pattern on the sheet. This will establish the pattern in relationship to the material’s grain direction. Since the flat pattern of the part is now a 3D object, there is the additional need to define which face is considered to be the top face. All this can be accomplished using the new Edit Flat Pattern Definition tool. This tool is found by right-clicking on the flat pattern icon in the browser, and allows the user to define an X-Y coordinate system for the part, as well as define a face that will be “up”.

As you can see in Figure 6, there is a second tab for Punch Representation on the Flat Pattern dialog. This brings us to another important element of sheet metal design work.

## ***Sheet Metal Punches***

A large portion of sheet metal parts are cut on CNC punching machines using shaped tools to punch specific patterns in the material. As part of the sheet metal module overhaul for R2008, a new system for handling punch data has been introduced. Punch features are now a special type of iFeature, which have added functionality to enable them to accommodate the particular needs of the sheet metal manufacturing environment. The iFeature authoring environment has also been streamlined to make the process of creating regular and punch iFeatures easier.



**FIG 6: ONCE THE FLAT PATTERN ORIENTATION HAS BEEN DEFINED, PUNCH ROTATION AND OTHER ANGLE-DEPENDENT PARAMETERS WILL BE REFERENCED FROM THAT ORIENTATION.**

The punch iFeature authoring tool is incorporated into the standard iFeature authoring tool. When the “Sheet Metal Punch Feature” option is selected, several new options become available. Here’s a run-down of the additions for sheet metal punch iFeatures:



**Center Point Required:** When the punch iFeature option is selected, the sketch for the punched cut will be required to contain a single center point. Other non-center points can be present, but the center point is used to establish the center of the punch iFeature when placing it on the sheet metal part. If an attempt is made to create a sheet metal punch iFeature without the center point, an error message will appear, and the punch will not be created.

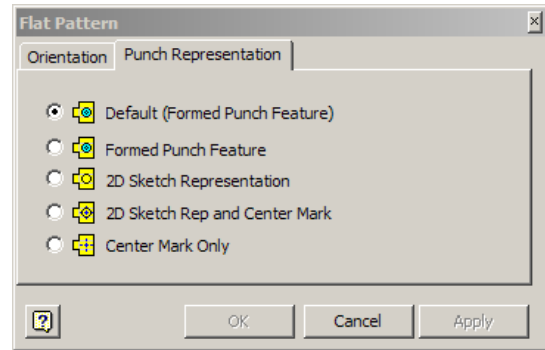


FIG 7: THE PUNCH REPRESENTATION TAB OF THE FLAT PATTERN DEFINITION DIALOG.

**Punch ID:** Provision has been added for specifying a text string to identify the punch. This is a common method of identifying punch tools in the manufacturing environment. This information is made available downstream in a number of ways. In this manner, libraries of punches can be more easily managed.

**Simplified Representation:** In some situations it is desirable to represent the punch on a drawing not as an actual cut in the unfolded part, but as a simplified representation of the punch. This can be useful in cases where there are hundreds of punches arrayed in a part, and the incurring computational overhead

of drawing them all is unnecessary. If desired, a second sketch can be selected which will be available later as an option for displaying the punch. As with the cut sketch itself, the

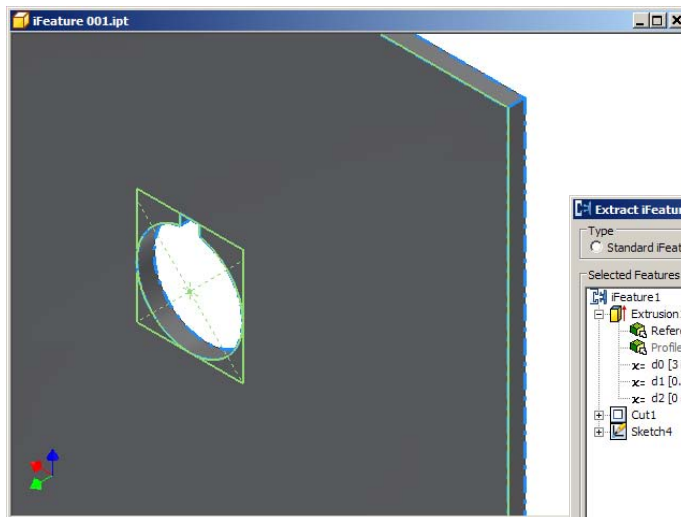
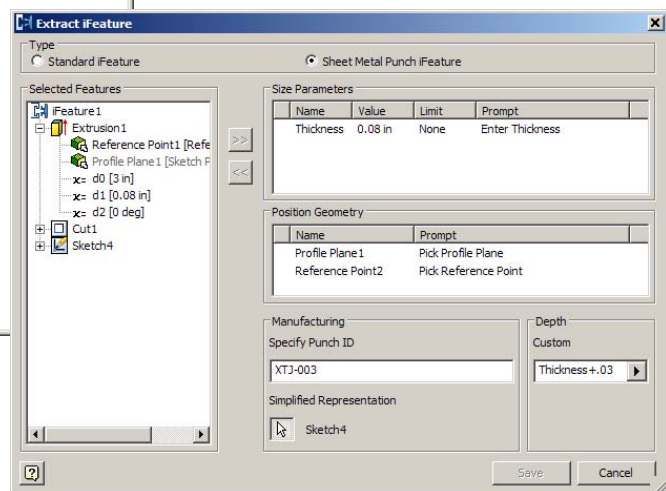


FIG 8: SELECTING THE SHEET METAL PUNCH IFEATURE RADIO BUTTON WILL MAKE THE SHEET METAL PUNCH FEATURES AVAILABLE, AND ALSO REQUIRE THE PUNCH SKETCH TO HAVE A SINGLE CENTER POINT.





representation sketch must contain a single center point. Obviously, the two centerpoints should be coincident. We'll look at how this option is used in the documentation section.

**Punch Depth:** The depth setting specifies how deep the punch tool throws into the part. Sometimes a specific punch throw depth is desirable for certain operations, and it can be specified here. If the iFeature is being created from a sheet metal cut feature, the Thickness parameter will be automatically brought over to the iFeature, and can be used as a reference in this field. In this way, the punch could be set to always travel a certain distance through (or into) the material, based on its thickness.

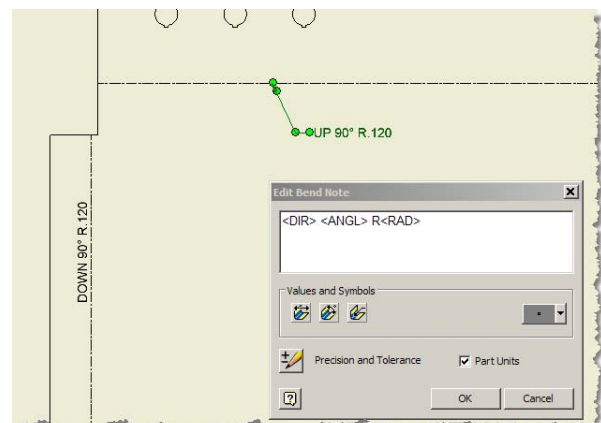
When placing punch iFeatures, the dialog box allows precise control over the configuration of the punch. Default options which were specified when the punch was created can be overridden when the punch is placed. If the punch is a member of an iFeature family, a specific version of the punch can be selected.

The folded model will always display the punch as a normal cut feature. For displaying the punch features in the flat pattern, one of four options can be selected (See FIG 7). The punch can be displayed as a normal cut, by using the alternate sketch representation if one has been defined, as the sketch representation with a center point, or as a simple center point alone. The default option is set in the sheet metal style, and can be overridden on the Punch Representation tab in the Flat Pattern Definition dialog box.

We'll be visiting punch functionality again in a moment when we look at Inventor 2008's documentation enhancements.

## **SHEET METAL DOCUMENTATION**

The evolution of the design/manufacturing environment towards a process that is free from the need for drawings has proved slower than some had predicted. It's quite likely that some day paper drawings will be as obsolete as the slide rule, but that time is not now. Designers still must produce drawings of their designs. Inventor's new direction for the sheet metal module has been accounted for in the implementation of new drawing tools for sheet metal components. Here's a brief look at the new tools within that context:



**FIG 9: BEND NOTES, ACCESSED FROM A FLYOUT UNDER THE HOLE NOTE TOOL.**

**Bend Notes:** The bend note annotation tool is found in the flyout under the hole note tool. Bend notes can be positioned directly on the bend line, or pulled off as a leader. The bend note can be set up in the style editor, and configured individually in a manner similar to hole notes.

**Bend Tables:** Bend annotation can also be formatted into a table. Within the standard table tool are options to select a part file to extract bend information. A bend table does not reference a drawing view, but goes directly to the source of the information: the flat pattern of the model.

**Punch Tables:** Punch information can also be arranged in the form of a table. In this case, the punch data can be referenced by the standard hole table, allowing punch and hole annotations to be grouped together. In the drawing view dialog (Component tab), check the option to recover punch centers. In the Edit Hole Table dialog, check the option to include recovered punch centers, and add punch information fields to the table using the column chooser.

Hole Table						
HOLE	YDIM	YDIM	DESCRIPTION	PUNCH ID	PUNCH ANGLE	PUNCH DEPTH
A1	7.02	6.86	8-36 UNF - 2B	-	-	-
A2	9.02	6.86	8-36 UNF - 2B	-	-	-
A3	3.03	7.97	8-36 UNF - 2B	-	-	-
B1	2.53	11.41	-	FHK-023	180	.15
C1	4.03	11.41	-	FHK-023	180	.15
D1	5.53	11.41	-	FHK-023	180	.15

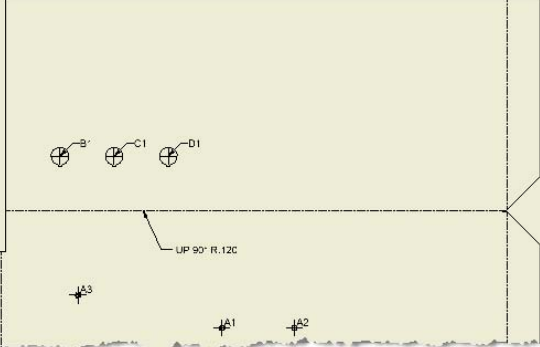
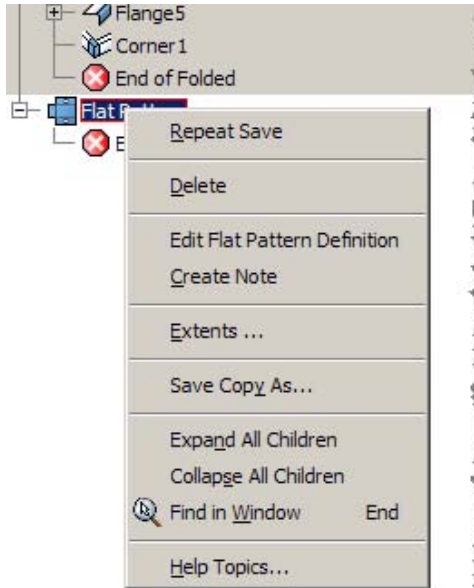


FIG 10: HOLE AND PUNCH INFORMATION SHARING A TABLE.

## FLAT PATTERN EXPORT

Until the day arrives when sheet metal parts are normally manufactured by directly querying the Inventor model, there will still be a need to export CNC-compatible flat pattern files. Release 2008 of Inventor has received a complete overhaul of its flat pattern export routine. In the past, the flat pattern export has been an extension of the old file export wizard. Now, a completely new tab-based export routine has been developed which is specifically targeted at sheet metal flat patterns. As with the flat pattern cleanup revisions, the goal of these improvements is to generate an exported file that can be read directly by CNC equipment without the need to tweak the file in a secondary application. This keeps control of the entire process within Inventor where it belongs.



**FIG 11: RIGHT-CLICKING ON THE FLAT PATTERN ICON IN THE BROWSER UNCOVERS A FLYOUT CONTAINING IMPORTANT FLAT PATTERN TOOLS, INCLUDING THE SAVE COPY AS TOOL, WHICH IS USED FOR EXPORTING FILES.**

The new flat pattern export routine is accessed by right-clicking the flat pattern icon in the browser and selecting "Save Copy As". This brings up a standard Windows File Save dialog. Three file export options are offered: SAT, DXF and DWG.

### **3D Flat Pattern Export**

Choosing the SAT file export option results in a 3D exported ACIS file of the flattened model. The option is presented to choose between various SAT file versions. The process is very straightforward.

### **2D Flat Pattern Export**

Choosing the DXF or DWG file export option brings up a new file export routine which is targeted specifically at sheet metal flat pattern exports. Instead of the traditional wizard, the new format is tab-based. The linear nature of the previous export wizards presented a series of choices one screen at a time, which sometimes made it difficult to

determine how the options on one page would affect the choices on the next. With a tab-based scheme, all the options are available in any order, so it's easier to make informed choices.

The help files within Inventor contain an excellent and comprehensive command reference for the flat pattern export dialog. There is really no need to duplicate it here by examining every option in detail. Suffice to say that several features have been added to the flat pattern export routine that are designed specifically to produce a file that is readable to most CNC cutting and bending machinery without the need for additional cleanup in AutoCAD or some other secondary application. Here are the highlights of the new functionality:

**Curve Simplification:** In the past, splines and arcs have been broken up into small straight segments for interpretation by CNC machines. Many modern machines can interpret these objects directly, so the option has been provided to export them intact. In cases where the segments are desirable, a control has been added to determine how closely the resulting segmented path conforms to the original curve. In many cases this can radically reduce the sheer number of tiny straight lines without significantly detracting from the accuracy of the cut. This results in much smaller and more manageable exported files.



**Layer Control:** Export to DWG or DXF files allows precise control over which layers are exported, and what they are named.

**Merge Profiles into Polyline:** For CNC machines that require profile shapes to be continuous polylines, Inventor gives the option to sense close boundaries and merge them automatically.

**Rebase Geometry into 1<sup>st</sup> Quadrant:** Selecting this option moves the geometry in the exported file into the positive quadrant of the X and Y axis.

As mentioned, exploring all the tabs of the flat pattern export dialog, and consulting the help reference for the various options is strongly encouraged.

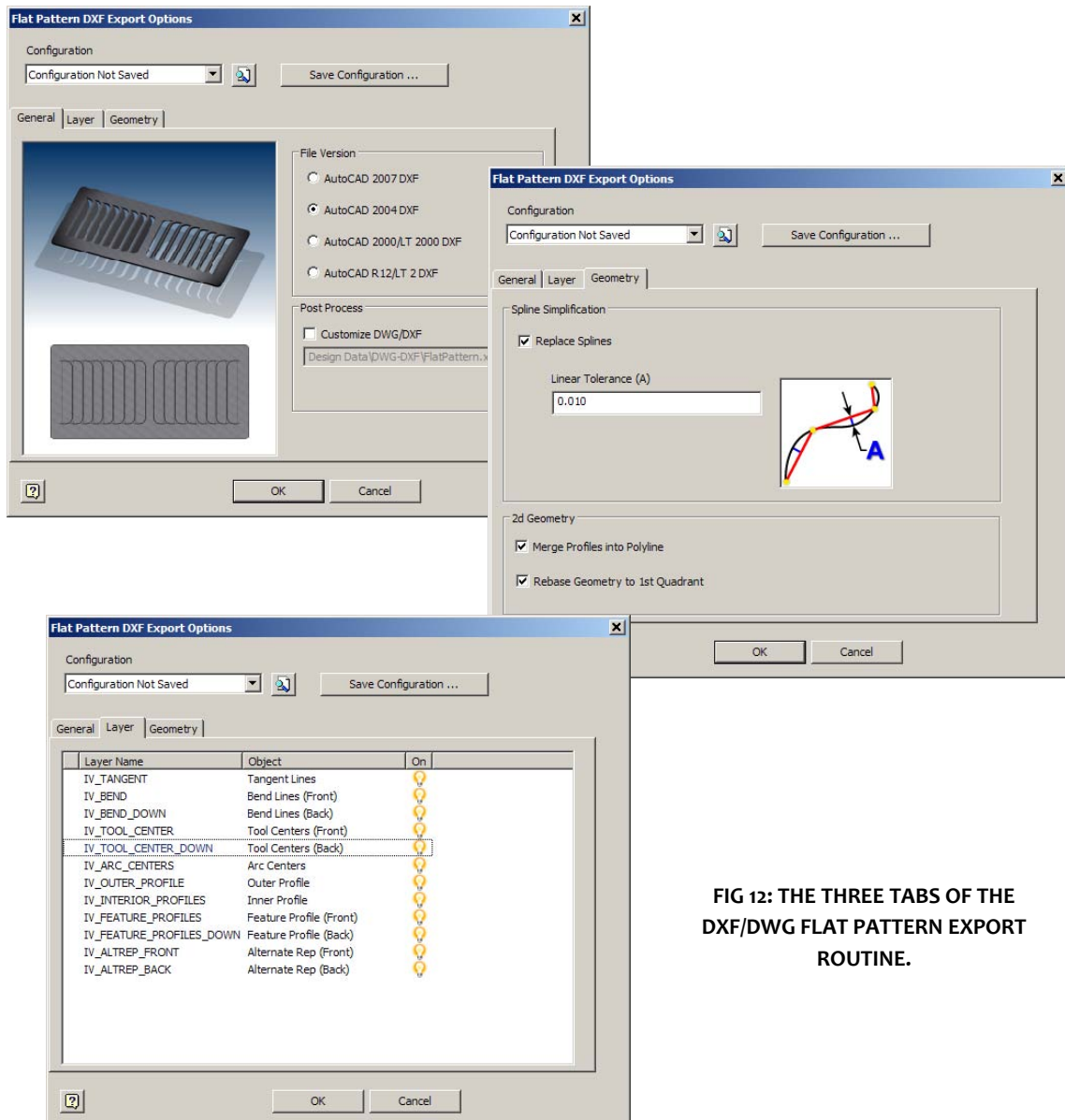


FIG 12: THE THREE TABS OF THE DXF/DWG FLAT PATTERN EXPORT ROUTINE.