Autodesk inventor sheet metal multibody

(SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇE (TURKISH))) ENGLISH (ENGLISH) (KOREAN) See more It was not possible to retrieve the original View X products table of contents and inventor 2016 by: Help In-Product View The multi-body parts environment is extended to include sheet metal parts. You can now use a top-down workflow to create multi-body sheet metal parts. Use the Make Components or Make Part command and open the derived components to create flat series for each part in a multi-body part file. The New Solid Body option is available in the following commands: Face Contour Flange Contour Roll Lofted Flange Split The following features support multi-body workflows: Flange, Angle Hem, Bend Cut, Bend, Tear, Punching Support for Zero Bend Radius are added to the following commands: Face Bend Flane Contour Roll Lofted Flange Hem Fold The amount of punch tool instances is added to the PunchTool dialog box , Geometry tab. When you convert a part to sheet metal, you are prompted to select a base face. This pick allows Inventor to measure the thickness of the material for you. After you select a face, the Sheet Metal Defaults dialog box appears. The measured thickness appears in the Thickness value box. Click OK to accept the measured thickness value. You can permanently close the warning message when you change the flat series. Some workshop tools were unable to use an exported DWG or DXF flat series file if a bend center area went through a hole in the bend. A new option is added to the Geometry tab in the Flat Pattern DXF or DWG Export Options dialog box. Select the new Trim Centerlines at Contour option on the Geometry tab to trim the bend centerlines to the edge of the cut. For more information View original x products and covered versions Autodesk Inventor Professional 2019 Maria Manuela Pinho Autodesk Platinum Partner 11 contributions 931 post 199 congratulations 77 solutions 2 ideas ScreencastIntermediate Sometimes it is difficult to project sheet metal parts, especially when it has complex shapes. A nice way to do this is to use multi-body modeling. Please take a look (SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇ E (TURKISH) (KOREAN) (SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇE (TURKISH) ITALIAN (ITALIAN) (KOREAN) (SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE))) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇE (TURKISH) ITALIAN (ITALIAN) (KOREAN) (CHINESE) SIMPLIFIED) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL ESPAÑOL TÜRKÇE (TURKISH) ITALIAN (ITALIAN) (KOREAN) See attached set of demo parts consisting of a part of 2 mm and 6 mm. ---Design Suite Ultimate 2021 Project Sites Around the World You've been detected as coming from . If applicable, you can view country-specific information, offers, and prices. Change country/language X Alt + g keyboard to turn grid overlay on or off In this project, you will design a bracket using the sheet metal environment within Inventor for an inkjet printer, while in the context of the larger assembly. Total project time of about 20 minutes. Autodesk Inventor software provides features that make it easy to create, modify, and document digital prototypes of sheet metal components A sheet metal part is often thought of as a part made from a uniformly thick sheet of material. If you design small objects, this material is often thin. However, in Autodesk Inventor you can use sheet metal commands on any project where the material is uniformly thick. Within the Autodesk Inventor design environment, a sheet metal part can be displayed as a folded or flat model. With sheet metal commands, you can open features and work on a model in a flattened state, and then rephrase features. You can create sheet metal parts from a template file. The sheet metal template file incorporates a set of rules. Rules determine some common attributes, such as material type and thickness, unfolding rules, space size, and so on. By changing a single rule, you can change the material of a sheet metal part from aluminum to stainless steel. A material change often requires changes to attributes that define bends and angles. Such modifications often require modifications to the workshop machines and equipment used to manufacture the parts. Like other parts created within Autodesk Inventor, sheet metal parts begin with a basic feature. The basic feature of a sheet metal part is often a single face of some shape to which other features (often flanges) are added. A complex design might use an outline flange or contour roll as an initial base feature. Some parts may use a loft flange as an initial feature. Unlike normal parts, sheet metal parts are always created from a uniformly thick sheet metal that is flat. This sheet is formed in the final part using various manufacturing techniques. In the sheet metal environment, you can create a bent model and unfold it in a flat pattern. The flat pattern is typically used to detail the Sheet metal controls used to work with flat patterns can provide critical manufacturing information. If a regular part created in Autoridesk Inventor has a constant thickness, you can convert it to a sheet metal part. The same applies to parts imported from other systems. (SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS PORTUGUÊS POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇE (TURKISH) ITALIAN (ITALIAN) (CHINESE)) (SIMPLIFIED CHINESE) ENGLISH FRANÇAIS (FRENCH) DEUTSCH (GERMAN) (JAPANESE) PORTUGUÊS (PORTUGUESE) POLSKI (POLISH) РУССКИЙ (RUSSIAN) ESPAÑOL (SPANISH) TÜRKÇE (TURKISH) ITALIAN (ITALIAN) (KOREAN) Do you find it frustrating to flatten complex forms of sheet metal in Autodesk Inventor frustrating? We received similar questions from some of our customers recently asking about flattening complex shapes, and a call we were able to resolve resulted in the creation of the item in the following image using the Autodesk Inventor 2016 multi-body sheet metal toolset. The activity involved creating a slide or slide with an elevation change of 2 m and a change of direction of 90°. What follows below is a short post describing how we identified the problem and a possible solution to it. Proof of possible outputBelow is a short video that demonstrates the output and shows a simple module created using Autodesk Inventor iLogic that demonstrates how well the parametric model behaves:Sticking to Autodesk Inventor Sheet MetalInitially I was puzzled about how I would approach this and had a couple of steps with sheet metal tools before having myself on the loft flange as my favorite weapon. When I faced another problem a couple of weeks ago, my colleague Paul Munford suggested that I break the problem into simpler segments, so this led me to a sketch that looked like this:Following the initial sketch, I then created a 3D sketch:Combining the 3D sketch and the elements of the initial 2D sketch I then made a swept surface on which I based the rest of the geometry :P who the resulting spiral profile is continuous, it means that I just needed to create a segment that can then be modeled parametricly along the path of the helical coil. Since this single segment uses loft flange as the main feature, the surface has been divided into triangular segments with Inventor creating a pressure brake fold on the face:And here magnified so that the shape of the loft flange is clearer:Now, since everything about this part is based on a well-known parameter (eagle-eyed cad-geeks among you will have noticed that I had a guided dimension in my first image ;-) , I can then use inventors rectangular series tool to guide this solid along my path:The final output of this now multi-body part is an assembly and since I made use of the multi-body sheet metal function available within Inventor 2016, I can now do it with relative ease:Since I made use of the Create Components tool within Inventor, the modeled body becomes its own file of But it remains connected to the parent part file :All member parts of that assembly are now able to be with flat models:to learn more about Autodesk Inventor or Autodesk Manufacturing Solutions, contact your local Graitec office today. Also, follow us on Twitter for production news, Autodesk, the future things and much more -- @cad4mfg. @cad4mfg.