Solidworks tutorial sheet metal pdf

SolidWorks UI SolidWorks UI is pretty simple and straight forward. There are 6 main interface areas that you usually work with. 1) Menu Bar - Top most applications, make new files, open files, save, print, un un un unsued, select, rebuild, file properties and options. [continue reading...] How to reflect a section in SolidWorks In this short SolidWorks tutorial I'll show you how to reflect your part and save time modeling other parts. First, get your section ready and make sure it's saved. Click [continue reading...] Create simple brackets with 2-line outline Can model this frame with two lines with SolidWorks, using thin extrusion features! First make L sketch and directly extrud the outline, look at thin features and set the thickness. This feature is simple and fast, it saves your time without having to outline offsets and outline sizes... Zoom Medium while you work with your part, assembly or drawing, you can quickly zoom in to fit your part in the work space by pressing the F key... Clear Up Crowded Sketch When you're in sketch mode, Solidworks automatically adds sketch relationships to your sketches. This helps you understand what relationships are recognized by Solidworks between neighboring relatives. Some time this outline relationship can fill you outline! How do I turn it off? Click >Skecth Relations Will Be Great! Modify the size with the combined unit You know you can modify the size with the mixing unit.. Have it a try.. SOLIDWORKS 3D allows you to quickly create sheet metal parts designs using a simple design process, saving you time and development costs, thanks to specific sheet metal features. We can use these features to create sheet metal designs with a number of different methods. We will focus on the flange method, where part of sheet metal is created in a formable state using specialized sheet metal features. To get started, we first want to enable the Sheet Metal tab on CommandManager. To do this, we just right-click any tab on CommandManager and select Sheet Metal from the drop-down menu. CommandManager Next, we'll activate command manager's Sheet Metal tab and click base flange/Tab tool, so the first tool on the right doesn't turn gray when starting a section. Base / Flange Tab To start this section, like any other tool, we need to select a plane and create a sketch. In this case, we will create a simple, open sketch to start our sheet metal section. Metal Part Sketch Once the sketch is confirmed a preview of the sheet metal section appears in the graphics area and property manager Flange appears on the left. Starting from the beginning of the asset manager, the first part is Direction 1. Similar to a basic extrusion feature, an end condition needs to be selected and a size also needs to be determined. Asset Management The next part of the asset manager is sheet metal meter, the only option in this section is to check the box that gives us a gauge. If the check box is selected a spreadsheet can be selected Drop-down menu to control the thickness of the sheet metal section. A spreadsheet is a spreadsheet that stores values for gauge thickness as well as bend radius. The default position for these tables is C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\Sheet Metal Gauge Tables If no measurements are used the thickness of the material and the bending radius can be entered into the sheet metal parameters. Here we can also choose to reverse the specified direction on which side of the sketch material is applied. Sheet metal parameters In the Bending Subsidy section, we can choose how SOLIDWORKS determines the position of the neutral axis as the flat sample calculation. By selecting K-Factor, Bend Allowance or Bend Deduction from the drag-down menu, a specific value can be entered. By selecting Bend Table or Bend Calculation, you can use Excel documents. (For more information on this: The last option in the property manager allows you to choose the type of relief that will be automatically added when the requested bend. Rectangle Tear Obround Once you have made your choice for each of these options and you hit the green check you are left with a metal plate part. You will now access all the sheet metal tools on CommandManager's Sheet Metal tab. You are on your way and can now start adding any other sheet metal features needed. For more information about SOLIDWORKS 3D CAD, visit Hawk Ridge Systems today! Here's a quick solidworks sheet metal guide. Sheet metal tools allow you to quickly create sheet metal parts designs using a simple design process, all saving time and development costs. Take a look at how this WorksSolidworks Sheet Metal GuideStep 1First Create a New Part.Step. Step 2Right-click the toolbar and activate Sheet Metal.Step 3Click on the top plane and then create a new outline. Step 4Now, sketch and use Smart Size to give a size to design. Step 5Click on the Sheet Metal tab, select Base Flange/Tab and provide 1 mm for sheet metal parameters. Step 6To bend the sheet metal part, outlining the lines on the section as shown in the picture. Step 7Bend the section by clicking Sketched Bend, or Insert >> Sheet Metal >> Sketched Bend. The section bends according to the outline lines. Step 8 In this step, we'll extend the edge with Edge Flange. Click on the right edge as it is shown below and extrud it by 30 mm.Step 9The Hem tool adds a hem to your metal plate part at a selected edge. In an open sheet metal section and in the graphic area, select an edge as shown. Step 10Repeat the last step again but this time try a different type of hem. There are specific sheet metal features that you can use to create metal bodies quickly. However, in some cases when the design requires certain types of type of metal, you can use non-plate metal feature tools, then insert bends or convert the sheet metal component. You might as well like: This is a quick Solidworks metal guide. Sheet metal tools allow you to quickly create sheet metal parts designs using a simple design process, all saving time and development costs. Take a look at how this WorksSolidworks Sheet Metal GuideStep 1First Create a New Part.Step. Step 2Right-click the toolbar and activate Sheet Metal.Step 3Click on the top plane and then create a new outline. Step 4Now, sketch and use Smart Size to give a size to design. Step 5Click on the Sheet Metal tab, select Base Flange/Tab and provide 1 mm for sheet metal parameters. Step 6To bend the sheet metal part, outlining the lines on the section as shown in the picture. Step 7Bend the section by clicking Sketched Bend, or Insert >> Sheet Metal >> Sketched Bend. The section bends according to the outline lines. Step 8 In this step, we'll extend the edge with Edge Flange. Click on the right edge as it is shown below and extrud it by 30 mm.Step 9The Hem tool adds a hem to your metal plate part at a selected edge. In an open sheet metal section and in the graphic area, select an edge as shown. Step 10Repeat the last step again but this time try a different type of hem. There are specific sheet metal features that you can use to create metal plate bodies quickly. However, in some cases when the design requires certain types of type of metal, you can use non-plate metal feature tools, then insert bends or convert the sheet metal component. You may also like: Welcome to SolidWorks Guide for Beginners, in this tutorial you will see how to use solidworks sheet metal model. This SolidWorks guide is entirely for beginners. It explains the basic features of sheet metal, how to use each feature with examples. Follow this to get more basic SolidWirks training materials. What is Sheet Metal? Metal plates are metal formed thin and flat pieces, which use plates with a thickness of less than 6 mm. It is one of the main and basic forms of metal work. You can cut or bend into different shapes using metal plates. The biggest feature of metal plates is that it is capable of being formed and shaped by no process. Each process is done after the metal and it gives different shapes or sizes at the end. Metal plates are formed by rolling applications, which consist of applications of large compressive force on long metal working pieces, through certain no rolls. I hope you are acquainted with the different rolling methods studied in mechanical engineering classes. (Wiki pedia) Typically, sheet metal thickness is mentioned as metal measurement and it ranges from 30 to 8 gauge. That means, the higher the gauge, the thinner the sheetmetal. Aluminium, copper copper, light steel, tin, nickel etc. are often used to create sheetmetals. The application of sheet metal It has extensive applications in different parts of the technique. It can be used to make bodies of cars, aircraft wings, electronic casings, home equipment casings, laptops, CPU cases, mobile phones etc. Bending or unbending metal plates are easy to do in real-life scenarios with the help of various machine tools and experienced mechanics. But, it is computer CAD programs like SolidWorks that are analyzed in nature. Therefore, you need to represent the process of bending analysis. To represent that, SolidWorks offers bend subsidies, bend deductions, k-factor etc. Bending subsidies That are subsidies for flat plate metals to bend to certain shapes. The curved part occupies a small part of the flat plate. So you have to explicitly mention the subsidies in SolidWorks. You have some bending subsidy values on the desired place or create bending subsidy tables for greater accuracy. Bend the deduction It's like the subsidy bends, but just reverses the bending subsidy. In the Bend deduction, you are deducting the bend section length from flat plate metal. Bend angle:- Angle represents the bend section. K-Factor:- It is the only value used to represent the bending or magic process on the bending angle, material thickness, bending radius, etc. The K-factor value will be obtained from the metal supplier or handbook. For example, for the value K copper steel and springs is 0.45. Material thickness: - That is the thickness of the sheet metal material used. Let us start the SolidWorks Sheet Metal Tools app. How to use SolidWorks Base Flange | SolidWorks SheetMetal #1 Base is the first and basic step used to start sheet metal modeling. It is applied one for each file section. You can create an open, single, or multiple closed profiles to create a base flange. Once you set the metal thickness and bending radius, it becomes the default value for additional sheet metal features added to the same base flange. Step 1: Open SolidWorks New Part File Open SolidWorks and create a new Part file. Then select any default plane (e.g. Top plane) and apply normal Give view. Step 2: Insert Base Flange Sheet Metal into File Part Go in the Insert menu, select Sheet Metal from the drop-down menu, and select Base Flange from the side drop-down menu. Insert -> Sheet Metal -> Base Flange Step 3: Create Sketch for The Base Flange It will be directed to the Sketch section. You must create an open (line), single closure (rectangular) or multiple closed outline profiles. Single Closed Sketch Profile - Rectangular here, I'll make the profile sketch rectangular. (This is just showing for example, no need to enter size).) After creating the sketch profile, click Exit the Sketch. It will go directly to the Real Estate Management Flange Facility. Here, you need to enter sheet metal parameters (thickness, bending subsidy, k-factor etc.). Once you enter these values, it will automatically set for other applications of sheet metal features on this base flange section file. Thickness = 3.0 mm K-factor = 0.45 Auto Relief ratio = 0.5 Apply these values and click ok (green marker) on asset management flange. Go to File and Save sheet metal flange for future application of other solidworks sheet metal feature tools. Open menu sketch profile - Line Line steps above up to 2 (outline of the base flange). Draw the line outline configuration as shown in the image above and apply all dimensions using the smart size tool. Then Exit The Sketch. It will go directly to the flange asset management facility. Enter the values below on the specified locations of the base flange property manager. Direction 1 Choose blind and enter 20 mm. Direction 2 Choose blind and enter 30 mm. Thick metal sheet parameters = 3 mm (set direction of adding material thickness to internal or external profile) Bend radius = 0.8 mm K-factor = 0.45 under bending automatic relief rate = 0.5 After entering all the values on the basis of flange asset management and click Ok. You create the metal sheet flange base section on SolidWorks using a single open sketch profile. Save the section file for future reference. How to Use SolidWorks Edge Flange | SolidWorks Sheet Metal Guide #2 In the base flange section, you see the first step to making metal plate design in CAD SolidWorks software. The Edge Flange feature adds flanges to the selected edge or more edges. The conditions for applying flanges Edge is that the right edges are linear. You can also add flange parameters such as flange length angle, bend position orientation, custom bending adjustment, relief type etc. in Edge flange asset management. You can follow steps 1 and 2 and the results are shown in the image below. You can then go to command manager or Insert Menu and select Edge Flange from the sheetmetal side drop-down menu.. Then, Select the edge to create the flange on top of it. Here, I select the Edge-1 and immediately the preview is available on that edge. In Edge Asset Management Flange, you can set the flange length, flange position and flange angle. There are five types of flange positions. They are external materials, internal materials, external bends, bends from virtual straps, tanguar to bend. After entering all the details, click the green tick button. How to Use SolidWorks Miter Flange | SolidWorks Sheet Metal #2 sheet metal miter flanges are like edge flanges that add one or more flanges on the base flange metal part. The miter flange is slightly different from the edge flange, because it needs an outline profile of straight lines or loops. And also outline the plane always perpendicular to the first edge, where you will create the Miter flange. There is no need for any confusion about the thickness of the sheet metal part, which is automatically selected from the base flange part. Miter Flange Video Tutorial Here I will create a miter flange on three edges using continuous lines (L-shaped lines or multiple continuous lines). Given that I have created the sheetmetal flange base partly shown. I have to create a flange miter the edges of this section. For that I chose a face as airplane sketch. (Condition: Airplane outline should be normal to edge first for outline). Select the Line sketching tool and start the line from the edge and create an L-shaped line. Select the sketch. Then, go to the Insert menu and from the drop-down menu select SheetMetal and Miter flanges shown below. Insert -> SheetMetal -> Miter Flange You can see a yellow preview showing the miter flange appearing on the edge selected Edge-1. Then, select edge-2 and Edge-3 clockwise and also view the preview below. You can also adjust the flange position from the Miter flange asset manager. There are three types of flange positions available and they are Internal Materials, External Materials and External Bends. Here I chose External Materials In addition you can adjust the Gap distance. It adjusts the miter flange distance between them. You can also apply offset for the start and end from the asset manager. Finally, click the green tick button. More SolidWorks SheetMetal Guides will publish soon... Stay tuned... Thank you, friends. Friends.