DOWNLOAD PDF SHEET METAL DESIGN PARAMETERS

Chapter 1 : SOLIDWORKS Help - Sheet Metal

Asked to answer, Sheet Metal Design Parameters are: Bends Counterbores Countersinks Curls Dimples Embossments Flanges Hems Holes Lances Notches Ribs Semi-Pierced Hole Slots Tabs Plating Welding, etc.

Contact Us 7 Must-Follow Guidelines to Strengthen Your Sheet Metal Design Be it the aerospace industry or the automotive industry, for the purpose of equestrian or airplane, sheet metal is the backbone of the modern industry. A strong backbone is a desideratum for a fine product. Sheet metal design plays a vital role in the making of a strong backbone. A designer must have a clear set of goals and design strategies established to ensure that part design is cost effective and can be easily manufactured. Function, attachment method, mechanical properties and manufacturing properties should be mulled over before moving well into the design cycle. The jarring difference between a slice of flat sheet metal and the body of a car is the testimony of the power of design. Depending on the previously decided objective, some would be labeled as good sheet metal design practice while others would be categorized as bad sheet metal design practice. Depending on the process adopted for bending, the K-factor in the area of bending is usually visualized. In the course of bending, the outer surface of the sheet metal witnesses more strain than the inner surface. Bending the sheet metal beyond a point would result in cracks on the outer surface. This point is named as minimum bend radius. A sheet metal designer should always conceive design ideas with minimum bend radius relating to the thickness of the sheet metal. In case of design intent with increased minimum bend radius, the sheet metal would need to undergo various processing such as polishing or grounding. Structured grooves, hole and slot Punching is an economical modus operandi for creating holes in a sheet metal. In such a case, manufacturing becomes easy requiring minimum and error-free punching, eliminating the prospects of breakage. An important design tip is to make provision for holes whose diameter is equal or more than the sheet metal thickness. The ductility and inner bend radius of a sheet metal is inversely proportionate to each other. The nature of several grades of sheet metal needs to be always taken into consideration in the course of designing. A Design for Manufacturability software often takes all these factors into consideration, proposing an accepted industrial standard accommodating the design idea of a designer. Fabrication alternatives Sheet metal, depending on their end use, often make use of variegated processes. Welding is one such process which may require rigorous grinding. In such a case, the designer must leave sufficient room to accommodate this fabrication option. Minimum Flange Width Flanges make the process of creation of a sheet metal part quick and convenient. In the course of conceptualizing a flange, the following should always be taken into consideration â€” Flange width must never be less than four times the thickness of the sheet metal. On failing to do so, the tool of choice will leave a mark on the sheet metal surface. Welding Most sheet metal designers include a provision for brackets in their design, not having weighed the available alternates. Is welding an absolute necessity or cutting the base material would lead to similar end results as well? Can mechanical fasteners help you achieve similar design goals? Efficient and simple design with minimal cost is the final goal for all designers! Wiping die bending Edge bending is one of the many processes of sheet metal. While this process has many advantages, failing to use this effectively can give rise to unwanted complexities. At the time of drafting a design with bend forming, make provision for angles less than 90 degrees as it would entail less cost and minimal use of complex equipment. While designers with their penchant for details strive for perfection, it may involve volumes of design rules and guidelines to ensure manufacturability and assembly. These guidelines can be hard to remember and time-consuming to check. Following most of the above mentioned guidelines, the presence of an efficient design for manufacturing and assembly software in the entire equation would ensure an error free end design. Further, the ever changing industrial practices demands an automated and standardized process to review all the design for manufacturing guidelines Learn more about what makes a great sheet metal design.

Chapter 2 : Jordan's Introduction to Sheet Metal - Google Docs

Sheet metal enclosures are fabricated by 'cold forming', where the metal is clamped and bent in machines called 'press brakes'. As a result, metal typically cannot be formed into a true 90 degree corner.

I thought some Inventor Sheet Metal would be a nice addition this week. The same Sheet Metal part when the Thickness variable is overridden from 0. Notice how the bend radii adapt. Inventor calculates and creates features based on variables that are stored in the Parameters, and editable through the Styles Editor. These are referred to as Sheet Metal Rules, which are distributed to all the Sheet Metal features such as Face and Bend as they are created. Sheet Metal Defaults are the driving force in the model, and direct which Rules and user overrides will be placed in the Sheet Metal Parameters. Sheet Metal Rules are a collection of instructions, that can and usually do fill the Sheet Metal Parameters as well as guide the feature behavior. Features can be driven globally from the Sheet Metal Parameters, or by individual user input. Sheet Metal Defaults Figure 2: These can be accessed through the Sheet Metal Defaults Editor: This keeps everything consistent. When I do have a special part thickness for example, then I select the best Rule set, and override the thickness in the defaults when needed. Sheet Metal Rules Rules are the master collection of factors that dictate Sheet Metal feature behavior in most cases. These are editable in the Styles Editor Dialog: Rules create a solid foundation for Sheet Metal features to be edited parametrically Figure 3: The Sheet Metal Rules in the Inventor Styles editor Sheet Metal Features and Parameters Features look to the defaults when being created, but can be overridden by the user as the features are being created. This is useful when non-standard bend radii, etc are required. Once created, the feature size values are stored in the Model Parameters, which are easily accessible through the Parameters Editor. This permits the entire design to adapt to changes, instead of only the default values. The Inventor Parameters editor. Notice the Sheet Metal parameter at the top. In the image above you should see the Sheet Metal Parameters that are read-only in the Parameters Editor. These are populated by the Sheet Metal Rules. Below that in the Model Parameters, numerous features were assigned named parameters based on the default Sheet Metal Parameters above, or factors thereof.

Chapter 3 : SOLIDWORKS Help - Using Sheet Metal Bend Parameters

Dear Friensd, In this video you will know about sheet metal design consideration, sheet metal design basic concept, you must have to know about basic concept of sheet metal design before designing.

Import into RefWorks 1. Introduction Currently, the demand for metal containers in the world is in the order of billion units per annum. Out of this, drink cans account for billion and processed food cans account for 75 billion. Reduction of material can be done by decreasing the wall thickness of the containers, whilst also retaining adequate strength to allow the container to serve its purpose, without fear of failure. The designer must meet these expectations, but also, must determine the material requirements and properties which are suitable for the food or drink being packaged. Traditionally, this process, as well as most metal forming techniques have been tested experimentally using trial-and-error or empirical methods, which are expensive and time consuming approaches, as dies, blank holders and punches need to be manufactured. The objective of the work is to successfully simulate the deep drawing process and validate the results, by making a comparison with previously published literature. The literature contains both experimental and numerical results which are in agreement with each other and therefore is being set as the benchmark to for comparison purposes. The incentive for doing this research is that deep drawing has come to a stage in the current industrialized world that requires the most efficient, low-cost, manufacturing route to be taken at all times. The successful simulation of the deep drawing process is very much dependent on successful modeling of the deep drawing process for numerical analysis. By making use of finite element analysis and statistical methods, the prediction of results such as the punch force, the blank holder force, the thickness distribution through sections of the metal and the lubrication requirements can be determined. This can significantly reduce the production costs, for higher quality containers by reducing the lead time to production and provides engineers the ability to promptly respond to market changes. In doing this, the level of knowledge in how various materials interact at the contact surfaces are enhanced, and the data for dealing with specific materials are also increased, which is another positive outcome. The first of these works is published by Colgan and Monaghan [2] which has taken a statistical approach, based on experimental design using orthogonal arrays to ascertain which factors most influence the deep drawing process. The geometries for the deep drawing process have been reproduced, and a critical comparison was made for the products. The authors explain that for the experimental component of the research, hydraulic cylinders were used to deliver the blank holder force. They also reported that a pressure gauge was used to measure the pressure in the cylinders in conjunction with a needle valve. Although providing adequate results, they suggested, it would be desirable to employ the use of strain gauges in this instance, so as to gain a higher level of accuracy and confidence in the values obtained. It was made apparent that the smaller the die radii, the greater the force on the blank, resulting in thinner wall thicknesses. It was also shown that the type of lubrication is very much affecting the force on the punch. The paper shows that under constant pressure the base of drawn cups remains at a constant value until thickness drops dramatically near the cup corners and follows exponential variation towards the outer edge on the cup flange. Taking into account the anisotropic properties of the material they concluded that forces exceeding 10MPa may result in tearing in the cup walls. A paper published by Waleed Khalid Jawad [4] confirms the previously stated point that increasing the punch radii can slightly decrease the punch load and vice versa. He investigated the effects of punch radii on maintaining the interfacial contact between the punch and the blank, and punch load on thickness through a section of the drawn cup, and finally predicted the resulting localized strain and stress distribution. He concluded that frictional forces act mainly through the edge of the punch in a shearing manner, with little effect on the flat section. Vladimirov et al [5] presented the derivation of a finite strain material model for plastic anisotropy and nonlinear kinematic and isotropic hardening. The work is applied numerically to the drawing of cylindrical and square shaped cups. They show that the numerical simulations, can be suitably extracted from the complex mathematical material models and that the phenomena of anisotropic properties can still be accounted for even with large deformations. They further go on to show the applicability of the mathematical work to account for the occurrence of earring during the drawing process. Fereshteh-Saniee and Montazeran [6] predicted forming load, by using various finite element types, and the strain thickness distributions are produced, as reported in [4]. The authors also show that the use of shell 51 elements of the finite element package give a much higher agreement with the experimental results, as compared with the results using visco solid elements. In making these comparisons, Fereshteh-Saniee and Montazeran use a formula derived by Siebel in [7] to produce an analytical solution to which the level of agreement between the various methods can generally be accepted. Meguid and Refaat [8] used a method of variation inequalities to develop an analytical method to model the frictional contact in elastoplastic models which undergo large deformations. The effect of nonlinearities arising due to either the geometry or the materials used in the model is handled by using the updated Lagrange formulation. Having arrived at a point where the material can be modeled to a very high degree of accuracy and the forming loads can be predicted to complete the drawing process, one may consider various frictional combinations at the contacting surfaces aiming at minimizing such forming loads, and hence cost analysis can be carried out for the manufacturing process of the container. This paper intends to approach the deep drawing process of thin walled, mild steel, cylindrical containers, by means of a finite element analysis. Simulation of Deep Drawing Process 2. Finite Element Model Figure 1 shows a sketch for drawing a circular cup. The process is described in the following steps: The circular blank is inserted between die and blank holder; the punch move downwards; the diameter of the blank decreases with the blank being drawn into the die cavity; the flange of the blank form the perpendicular wall of the cup and the center part of the blank form the bottom of the cup. The important dimensions, Figure 2 , of the blank, die, punch, and blank holder are shown in Table 1. Sketch of the drawing die assembly Download as.

Chapter 4 : Autodesk Inventor Distributed Parameters in Sheet Metal | Design & Motion

Download the sheet metal design guide for design for manufacturability best practices on hems, countersinks, holes and slots, bends, and more. The guide will continuously be updated from engineering feedback, so if you are looking for a specific feature to be highlighted let us know.

You must be in the Generative Sheetmetal Design workbench. Creating Sheet Metal Parameters 1. Click the Parameters tab. Enter a value in the Thickness text box. Enter a value in the Default Bend Radius text box. If desired, click the Sheet Standards Files to select company defined standards. Options Bend Extremities Click the Bend Extremities tab or click the icon to display the different types of bend extremities. Minimum With No Relief The bend coincides with the common area of the supporting wall. A corner relief is not created with this option. Square Relief The bend extremity has a square relief added to it. You can enter different values in the L1 and L2 text boxes. Round Relief The bend extremity has a round relief added to it. Linear In the unfolded position, the bend is split by two planes going through the corresponding limit points. Tangent The bend edges lie tangent to the edges of the supporting wall. Maximum The bend is located between the furthest opposite edges of the supporting walls. Closed The Bend is formed by the intersection between the bends of two supporting walls. The closed bend extremity lies on the surface of the encountered bend. Flat Joint The Bend is formed such that the two bends are joined in the flat view. Bend Allowance Bend allowance is automatically calculated by the K factor. The K factor is considered the neutral axis, which is a line somewhere in the thickness that is not stretched nor compressed when a bend occurs. This ratio represents the location of the neutral sheet with respect to the sheet metal thickness and the bend radii defined in the parameters tab. Many different factors can make up the K factor. Change the default by clicking in the grayed out area of the value, right-click and select Formula Deactivate. CATIA applies the corner relief after the wall with the intersecting bend has been applied. Tips Pay attention to which corner relief applies to which bend. You might delete a wall with a blend that has no corner relief and achieve the following result.

Chapter 5 : Autodesk Inventor Sheet Metal Parameters

A Sheet-Metal1 feature in the FeatureManager design tree indicates a sheet metal part. The Sheet-Metal1 feature contains the default bend parameters. Parent topic Sheet Metal.

Chapter 6 : Inventor Sheet Metal Design - Applied Engineering

Once you activate the base flange tool, you have an available list of parameters you need to specify to determine how sheet metal is going to stretch. In the picture above, I specified the parameters as follows: Thickness is T = mm, Bend Radius is R = mm and K-Factor is set to

Chapter 7 : Effect of Die Design Parameters on Thinning of Sheet Metal in the Deep Drawing Process

If you uncheck this option in the Sheet Metal Defaults, the option to use another parameter is enabled, and the 'List Parameters' option appears on the right of the Thickness field. Additionally, this unlocks the parameter field in the Parameter Editor, where you can manually type the derived parameter name, or simply use the 'List.

Chapter 8 : Editing the Sheet and Tool Parameters

A sheet metal's ability to withstand stress in a flexure test is an essential facet of sheet metal design. Depending on the process adopted for bending, the K-factor in the area of bending is usually visualized.

Chapter 9 : 7 Must-Follow Sheet Metal Design Guidelines to Strengthen your Design

Link features to sheet thickness Sheet Metal Parameters: for airflow in both a plastic or sheet metal design.