



**CHAPTER
THREE**

INTEGRATING DESIGN AND MANUFACTURING WITH SOLIDWORKS:

DESIGNING FOR MANUFACTURABILITY: PART I





CHAPTER THREE

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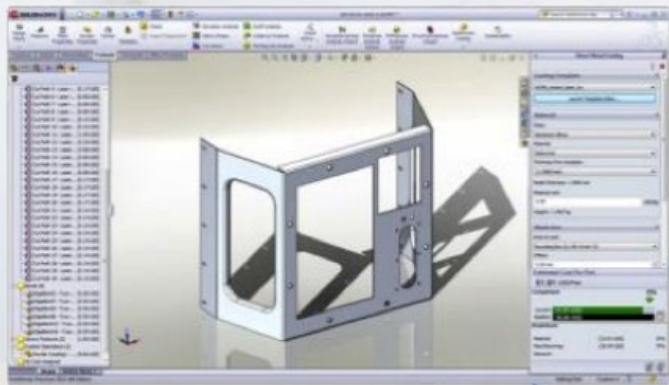


CHAPTERS 1 AND 2: RECAP

In the first of our series of eBooks on Integrating Design and Manufacturing with SOLIDWORKS®, we discussed the advantages offered by adopting an integrated design and manufacturing solution. In the second eBook, we discussed the advantages offered by SOLIDWORKS CAD.

In this third eBook, you will learn about the SOLIDWORKS solutions that enable you to determine the manufacturability of your products while you're still in the design phase, which can save you significant time to market and cut unnecessary costs and work out of your product development process.





SOLIDWORKS COSTING FOR AUTOMATED COST ESTIMATION

Provides instant, accurate, and repeatable manufacturing estimates directly from any 3D CAD model.

For manufacturers and designers

Manufacturers and estimators can use SOLIDWORKS Costing to create accurate, repeatable quotes based on customizable inputs reflecting actual labor costs, equipment costs and material costs. Designers, project managers, and purchasing can use SOLIDWORKS Costing to ensure their new designs are on target from a cost perspective by constantly monitoring current cost to manufacture their designs versus their original target costs.

Automatic manufacturing operation recognition

SOLIDWORKS Costing automatically recognizes and calculates manufacturing time, labor, setup, and materials costs for processes such as:

- **Machined parts**—milling, drilling, turning and mill/turn
- **Machining from custom**—shaped stock—machine castings, forgings and other custom stock
- **Sheet metal parts**—laser/waterjet/plasma cutting, punching, bending, welding
- **Plastic, cast and 3D printed parts**—injection molding and casting, as well as 3D printing
- **Welded structures**—welded and/or structural fabrications, welds and machining operations
- **Custom manufacturing operations**—painting, anodizing, deburring and grinding, shipping, ERP entry, etc. are automatically or interactively calculated

Operation set up and part load/unload calculations

Also included are calculations for the number of operation setups and the time to load and unload jobs into fixtures. Discounting and markup are also included, as well as the ability to set quantities of parts being produced.

Cost any 3D model, not just for SOLIDWORKS models

SOLIDWORKS Costing works with native SOLIDWORKS models as well as models from other 3D CAD tools as well as IGES and STEP. Users can define different part materials, make design changes, or change where the designs will be manufactured and instantly see the effects of these decisions on the unit cost of the part.

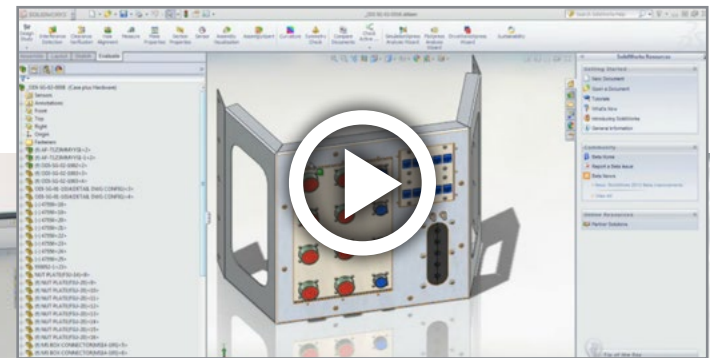
Designers, project managers and purchasing can use SOLIDWORKS Costing to ensure their new designs are on target from a cost perspective.

Easily customize your manufacturing settings and environment

Users can create custom [Costing Templates](#) which represent their actual manufacturing environments. Multiple templates can be created to represent different manufacturing facilities or processes. Templates are simple to create and understand because they are simple spreadsheets. Default templates are included with SOLIDWORKS Costing, giving users a starting point.

Output customizable cost reports and quotes for internal and external use

Costing Reports can be output automatically in both Microsoft Word and Excel formats. These reports, like the Costing templates, are completely customizable so that users can create the precise automated cost deliverables they desire—whether for external customer facing quotes, or internal production planning and costing.



VIDEO: SOLIDWORKS Costing

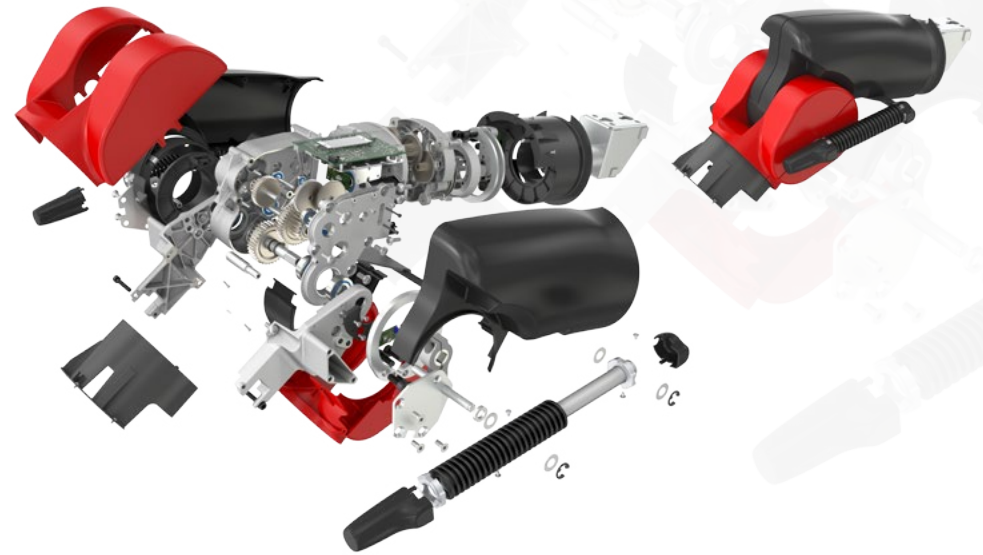


PLASTIC/CAST PART DESIGN, MOLD DESIGN, AND INJECTION MOLD FILLING SIMULATION

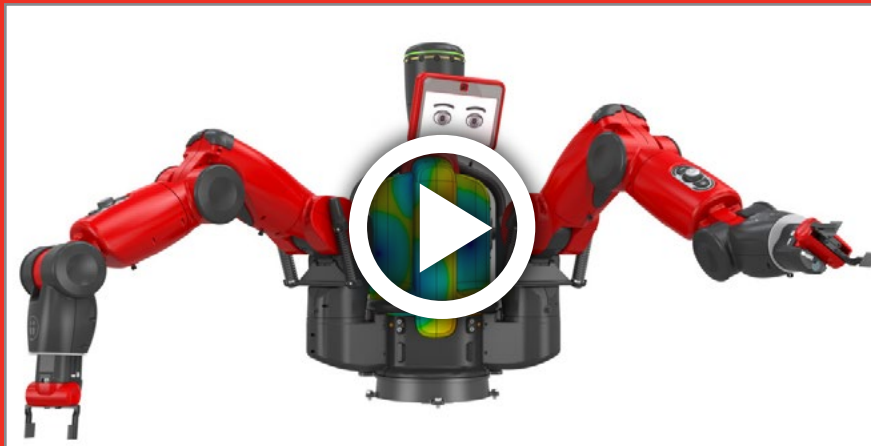
Provides the tools for both plastic/cast part designers and mold designers.

Plastic and cast part design

SOLIDWORKS 3D CAD help users develop [plastic and cast part designs](#) that meet requirements for product performance and manufacturability. A broad set of design tools enables the creation of simple or complex plastic and cast parts and ensures that designs can be successfully molded and manufactured.



See it in action: speedy creation and validation in 3D



VIDEO: Plastic Part Design

Plastic and cast part design tools include:

- Shell tool for ensuring uniform wall thickness
- Draft tool to add proper draft for mold extraction
- Fillet tool to create even the most complex fillets
- Tools to check draft, undercuts, thickness and surface characteristics like curvature
- Rib tool for adding stiffening ribs
- Complex surfacing tools to create and edit complex geometry, including stylish, curve-continuous (C2) surfaces
- Special purpose tools, including wrap tools
- Design for hardware-less assembly using Fastener Features, including Mounting Boss, Snap Hook & Groove, and Lip/Groove
- Vent design tool to plan for air flow cooling
- Automatically estimate plastic and cast part manufacturing costs as you design and/or create automatic quotes using SOLIDWORKS Costing functionality

SOLIDWORKS enables product designers and mold makers to easily incorporate design changes throughout the development process, right up to final manufacturing. The software—which can be used for plastic, cast, stamped, formed, and forged designs—fully integrates product design, mold design, and validation in one package, saving time, reducing costs, accelerating the product development process, and increasing productivity.

SOLIDWORKS associativity allows changes made to the design model to automatically flow throughout all the downstream models and documentation, including the mold and electrodes, as well as all the 2D drawings created from these 3D models. If SOLIDWORKS Gold CAM Partner software is used to create the machining tool paths for the mold and electrodes, those tool paths will update as well.

There is no need to manually track and edit individual files. All designs—parts, the mold tool, electrodes, drawings, and even the NC data—stay current and accurate.



“SOLIDWORKS has allowed us to reduce design cycle times significantly, which saves time and money, while simultaneously improving quality.”

— Mike Derus, Lead Engineer,
THE OUTDOOR GROUP LLC

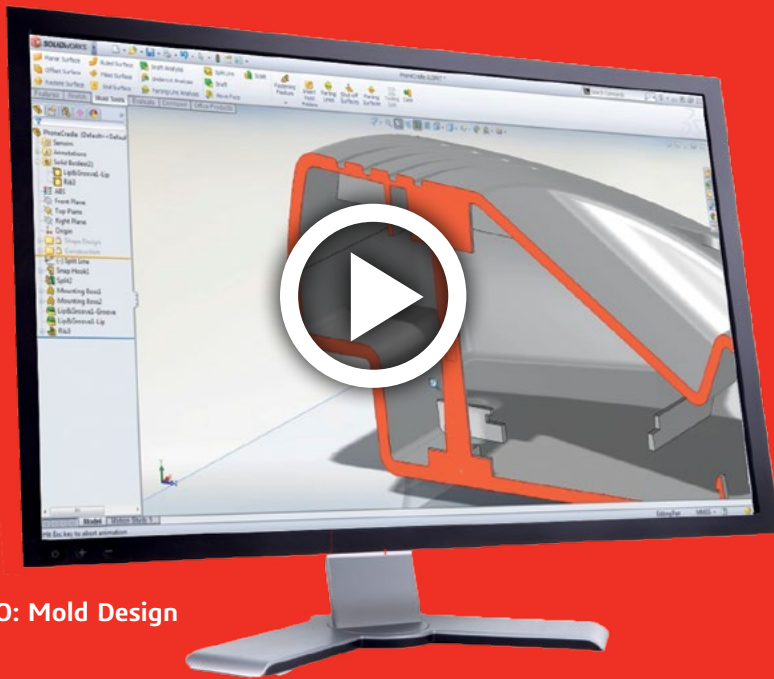


Mold Design

Complex mold designs change constantly during part development. Constant import, export, and repair of data, as designers move models between part and mold design, can consume valuable design time and introduce errors and inaccuracies due to data translation. As a result, mold designers typically wait until the part design is “frozen” before they start tooling.

SOLIDWORKS provides an advantage with design and validation of parts and tooling in one system. Designers can validate that parts can be molded successfully, including checking for draft, thickness, and undercuts to ensure the correct geometry for molding. Additionally, designers can use SOLIDWORKS Plastics to simulate mold filling and determine the best gate locations, whether the part will fill correctly, where weld lines might form, and possible sink locations.

LEARN ABOUT THE REAL-TIME VALIDATION CAPABILITIES OF SOLIDWORKS FOR MOLD DESIGN

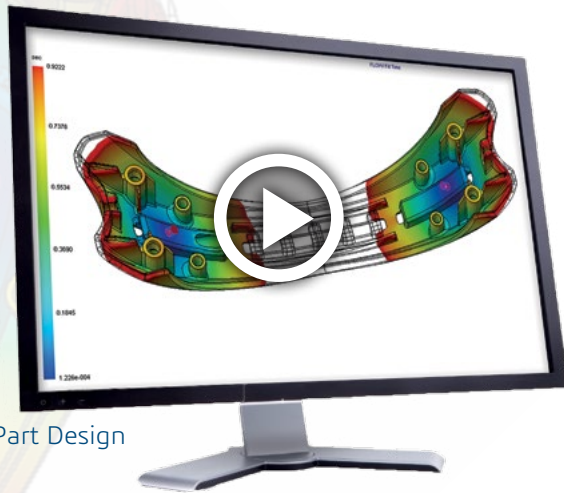


VIDEO: Mold Design

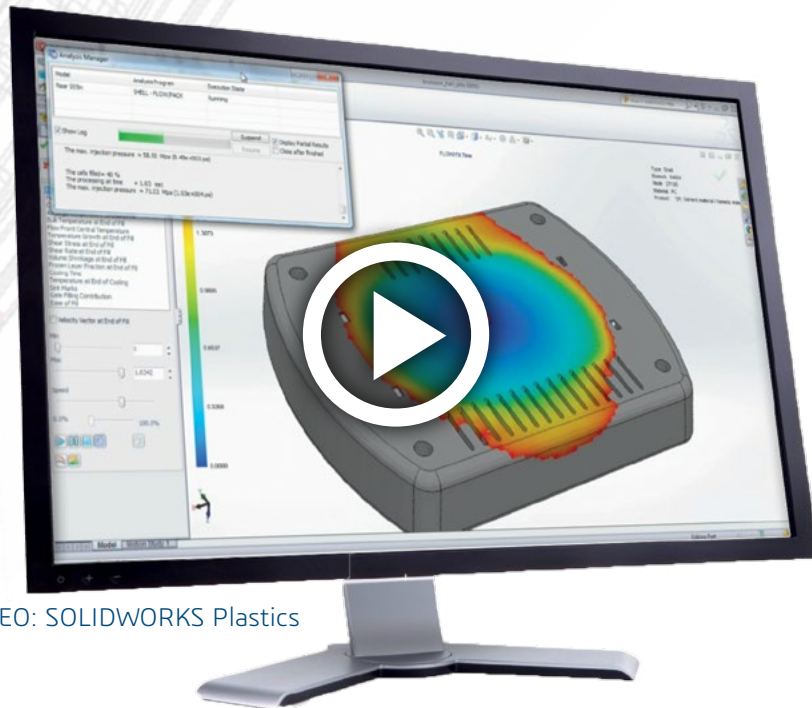


“I was attracted to SOLIDWORKS because I felt that using a feature-based parametric platform would be the most productive way to get my job done.”

— John Kreutzberger, Owner,
JK MOLD DESIGN



VIDEO: Plastic Part Design



VIDEO: SOLIDWORKS Plastics

SOLIDWORKS mold design capabilities provide complete plastic part and advanced surfacing design tools.

Key SOLIDWORKS mold design capabilities include:

- Complete plastic part and advanced surfacing design tools
- Geometry Import, Repair, and Healing tools
- Draft Analysis, Undercut Analysis, and Thickness Analysis tools
- Parting line tools
- Parting surface tools
- Automated “fill surface” capability for easy shutoff surface creation
- Core and cavity splitting tools
- SOLIDWORKS 3D mold bases available online and directly from major mold suppliers
- SOLIDWORKS configurations to create customizable configurations of molds and mold components
- Compare Part tools to visually see in 3D what geometry was added or removed when design changes occur
- Mold filling simulation
- Production-quality mold drawing creation tools
- Seamless integrations with several industry leading CAM products allowing NC tool paths to automatically update with part design and tooling changes

Injection Molding Fill Simulation and Analysis

[SOLIDWORKS Plastics](#) brings easy-to-use injection molding simulation directly to the designers of plastic parts and injection molds, as well as advanced CAE analysis. It simulates how melted plastic flows during the injection molding process to predict manufacturing-related defects on parts and molds. You can quickly evaluate manufacturability while you design, to eliminate costly mold rework, improve part quality, and accelerate time to market. A Results Adviser provides troubleshooting steps and practical design advice to help diagnose and avoid potential problems.

SOLIDWORKS SHEET METAL CAPABILITIES

Provides functionality for design as well as data outputs needed for manufacturing.

Gain the flexibility to quickly and cost-effectively create [sheet metal part designs using SOLIDWORKS](#) thereby accelerating the design process, saving time and development costs, and increasing productivity.

Because many industries require a wide range of sheet metal parts to house or enclose their designs, SOLIDWORKS provides a flexible design approach. SOLIDWORKS enables you to also automatically flatten your sheet metal design with bend allowance and document it for manufacturing. Flattening provides a first check of manufacturability for sheet metal parts. In addition, a flattened 2D drawing view can be created containing all needed manufacturing information.

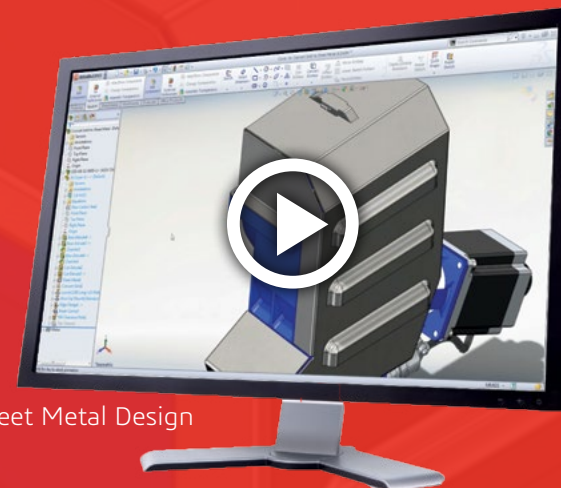
Finally, you can also output a DXF of a sheet metal part at a 1:1 scale, directly from the 3D sheet metal model, saving time and reducing errors in going to manufacturing.

With SOLIDWORKS you can:

- Design SOLIDWORKS 3D solid sheet metal parts and assemblies
- Convert SOLIDWORKS 3D models or imported CAD models directly to SOLIDWORKS sheet metal models and automatically input your own bend radius, thickness, and K-factor
- Design around a group of parts in an assembly
- Flatten your 3D sheet metal part to generate flat patterns that include bend compensation for manufacturing
- Generate Base, Edge, Miter, and Swept Flanges
- Generate bends, including Lofted Bends, Sketched Bends, and more
- Bend allowance/bend deduction information to ensure proper blank sizing for manufacturing
- Use Forming Tools to create features like ribs, louvers, lances, embosses, and extruded flanges
- Add weld details to sheet metal parts on models or drawings
- Punch tables, hole tables, bend tables, and weld tables
- Automatically estimate sheet metal part manufacturing cost as you design and/or create automatic quotes using SOLIDWORKS Costing functionality



VIDEO: Sheet Metal Documentation



VIDEO: Sheet Metal Design

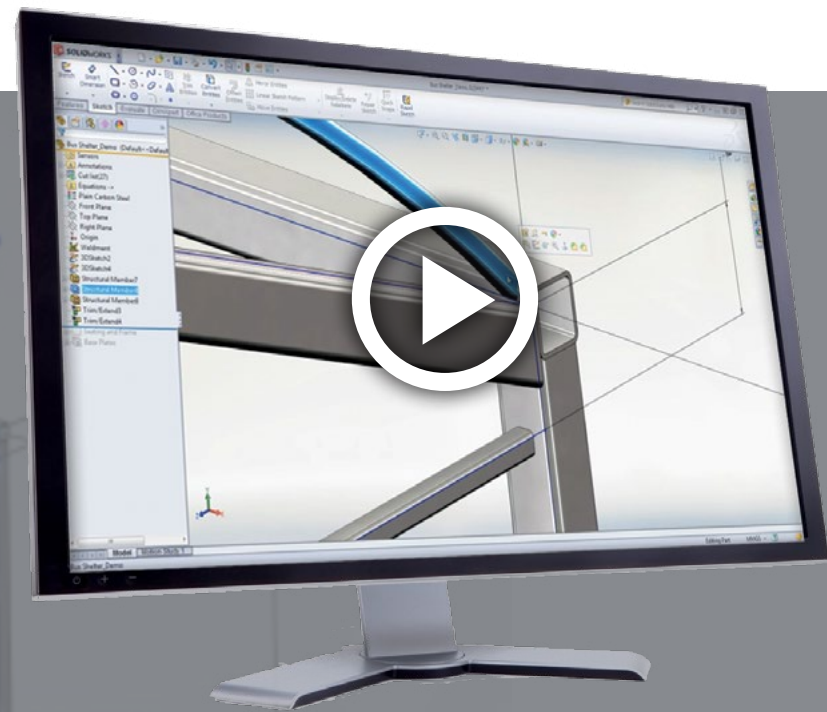
SOLIDWORKS WELDMENTS FUNCTIONALITY Provides outputs for welded structure design and manufacturing.

Use SOLIDWORKS to streamline the design and manufacture of [welded structures](#), frames, and bases that form the backbone of products developed in many industries. Quickly create designs that have extrusions and generate cut lists and bills of materials (BOMs) needed for manufacturing. SOLIDWORKS accelerates the design process, saving time and development costs, and increasing productivity.

SOLIDWORKS weldments design functionality enables you to design a weldment structure as a single multi-body part, sketching the basic framework, creating structural members with groups of sketch segments, and adding elements like gussets and end caps to complete the structure.

With SOLIDWORKS Weldments Functionality you can:

- Work from a library of pre-defined structural shapes
- Generate 3D sketch lines for structural profiles to follow
- Automatically trim structural members at intersections
- Add stiffening plates, gussets, and end caps
- Add optional weld beads to the design
- Generate drawings, BOMs, cut lists, and other manufacturing documentation
- Automatically estimate welded structure manufacturing cost as you design and/or create automatic quotes using SOLIDWORKS Costing functionality



VIDEO: Weldments

ADVANCED SURFACE FLATTENING FOR BLANK/PATTERN SIZING

Allows flattening of complex geometry in order to determine material size for manufacturing.

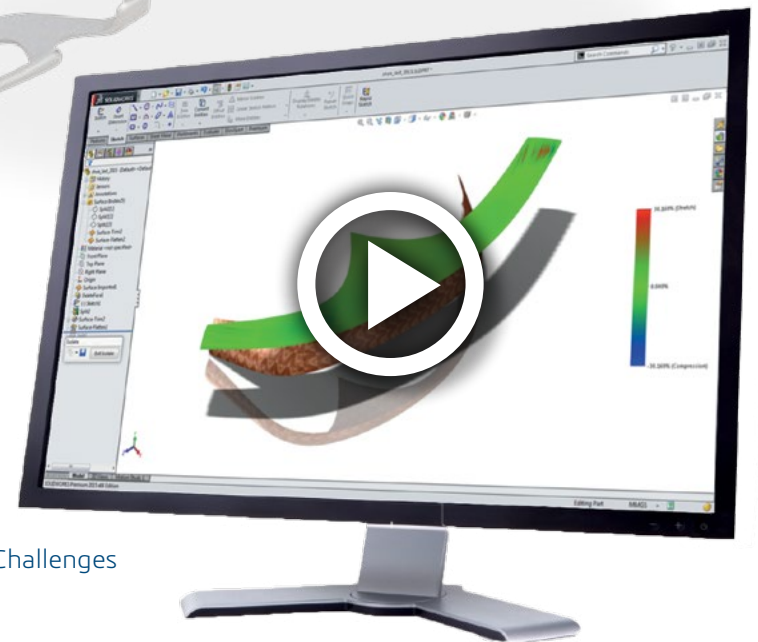
Often products, such as leather goods, sports equipment, footwear, and products made from composite materials like carbon fiber and fiberglass, have complex shapes, but are manufactured from materials that start out as flat stock, such as cow hides, nylon, cloth, and flat composite sheet. Therefore, it is necessary to accurately determine or estimate the size and shape of the surfaces in their flattened state to be able to cut the material from the flat sheet.

Automated and interactive surface flattening and data export

SOLIDWORKS has always provided the ability to automatically flatten certain types of geometry automatically. The SOLIDWORKS Flat Pattern feature for sheet metal parts provides automated flattening per sheet metal K-Factor and Bend Tables. However, when geometry is more complex, such as with nonanalytic surfaces, more control is often required to determine how the surface should be flattened.

SOLIDWORKS Advanced Surface Flattening functionality provides control and flexibility to easily flatten these more complex surfaces.

You can automatically flatten the surface and interrogate an interactive deformation plot to graphically view stretch and compression concentrations that can occur when flattening some surfaces with high curvature. Split lines or cuts can then be added to allow for relief of stresses in these areas. You can also select “control edges” to guide the flattening process to maintain length and shape of specific edges, which is particularly helpful for more highly contoured surfaces. These flattened surface outlines or patterns can then be exported for material cutting in a variety of export formats.



VIDEO: Design Challenges

**DOWNLOAD
NOW**



Don't miss the next in our series of Integrated Design and Manufacturing eBooks.

CHAPTER 4

DESIGN FOR MANUFACTURABILITY, PART II

Download the next in our series of eBooks, Designing for Manufacturability Part II, which will cover the following SOLIDWORKS-integrated solutions:

- SOLIDWORKS Inspection
- SOLIDWORKS Composer
- 3D Printing
- Electrical
- Cable Routing and Tubing