Sheet Metal Design

21, April 2023

Version 2023

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1 Sheet Metal Design

1.1 Introduction

CADdy++ Sheet Metal Design is an additional module to *CADdy++ Mechanical Design professional*, which allows parametric 3D sheet metal designs to be created easily and efficiently. The program is designed and programmed according to customer specific presettings.

In addition to the enhanced 3D functionality, knowledge bases also support *CADdy++ Sheet Metal Design* during the actual design phase. An Access database exists for the sheet metal design as a data basis for the verification of manufacturing conditions. This database contains all data concerning manufacturing operations (finisher), material, bending machine, cutting machine and bending tools as well as their relation to each other. Additional data, such as general addresses, manufacturing services, additional material properties, etc. can also be included in this database. This additional data acts purely as an information basis and is not taken into consideration during the verification stage of the sheet metal design.

All data is defined according to specific requirements and alternatively can supervise the design and bending simulation so that they can react to any possible errors that may occur during the design process.

1.2 Installation

Installation of *CADdy++ Sheet Metal Design* is carried out within the framework of the *CADdy++ Mechanical Design professional* installation. Information concerning this can be found in the respective manual.

ODBC connection

All data concerning *CADdy++ Sheet Metal Design* is saved in the Access® database ***CADdyMABK.mdb***.
After installation, this file can be found in the **..*CADDY*MA\Program\Database** folder.
CADdy++ Sheet Metal Design accesses this file with the use of a temporary ODBC connection, which is set up once the program is started. This ODBC connection is given the name

CADdyMABKIntern3D. This connection is removed again once the program is quit. If you wish to be able to access the database in another folder, e.g. on a network drive, your system must be given the exact location and name of the database driver.
This connection between data source, driver and the *CADdy*++ Sheet Metal Design program is made via the ODBC interface.

In order to set this up, please proceed as follows:

1. Copy the CADdyMABK.MDB database into the required folder.
2. Select – beginning in the ***Start*** menu - the commands ***Settings, Control Panel***.
3. Activate the ***ODBC Data Sources***.
4. Here, define a new data source by clicking on the command button ***Add***.
5. In this dialog box, tag the required driver ***Microsoft Access Driver (*.mdb)***.
6. In order to allocate a database source to the selected driver, activate the command button ***Configure***.
7. In the ***Data Source Name*** text box, enter ***CADdy*MABK3D**. This is the *CADdy*++ *Sheet Metal Design* internal name for the database – therefore please take note of the correct spelling.
8. Using the ***Select*** command button, now select the folder that contains the ***CADdyMABK.mdb*** database file.
9. Tag this file and then exit the dialog box using *OK*.

The connection described here must be carried out on each computer that is to have access to the *CADdy*++ *Sheet Metal Design* database.

To ensure that all *CADdy*++ *Sheet Metal Design* users have access to this file, the network administrator must allocate respective read and write rights.

2 Settings for Sheet Metal Design

2.1 Knowledge base

All data relevant for the verification of a sheet metal design can be entered in the Access® database ***CADdyMABK.mdb***. After installation,

this can generally be found in the **..*CADDY*MA\Program\Database** directory.

Open this file using Microsoft® Access (min. requirement: Office 2000) in order to enter the required data in the respective tables.

For the enhancement or individual adaptation of the database, please contact DataSolid GmbH. We recommend that personal enhancement to the knowledge base should only carried out by experienced Microsoft® Access users.

Specific settings for the actual sheet metal design e.g. sheet thickness, bending radius, etc. are carried out directly in *CADdy*$^{++}$ *Mechanical Design*.

2.2 Parameters

Settings for your respective sheet metal design can be made directly in *CADdy*$^{++}$ *Sheet Metal Design*.

After loading *CADdy*$^{++}$ *Sheet Metal Design* using the menu ***Extras/Sheet Metal Design***, change to the ***Parameters*** option card if desired.

Sheet Properties
You can define your general sheet parameters under this heading:

- *sheet thickness*
- *bending radius*
- *material*
- *manufacturing profile*

Please note
Entries made in the *material* text box have an affect on the verification of the rules defined for sheet metal design. In this text box, enter one of the materials defined in the knowledge base. The manufacturing profile is a password for all or a selection of manufacturing relevant data (machine, material data, etc.) of a sheet metal finisher. In this text box, enter one of the finishers defined in the knowledge base.

Geometry
Determine the ***indentation thickness*** and the ***indentation extension***.

Options
The following two settings can be made under this heading:

Lines in current WP:

with copy – if a new workplane is generated whilst defining a new plate to the set of joins, the selected line will also be copied into the new workplane. In this way, you can always refer to the existing 2D geometry when designing the new plates.

without copy – if a new workplane is generated on the selected line, the line is not also copied in the new workplane.

Current rule

automatic: supervises your sheet metal design during the design and/or bending process according to the defined rules. If the  command button at the bottom border of the dialog box changes its colour from green to red, this indicates an error. By clicking on this command button, the message is quit and the command button displays green again. The error can be displayed using the ***Error Status*** command button on the option cards *Join* and/or *Bending Simulation*.

load command: only displays errors in accordance with the defined rules once you have started verification of your design.

Bending zone text

text size: defines the text height of the bending zone numbering. From this text, the order of the bevels can be read.

3 Sheet Metal Design

3.1 General Procedure

In order to load *CADdy++ Sheet Metal Design*, please proceed as follows:

1. Activate a 3D window in *CADdy++ Mechanical Design professional*
2. Select the menu command *Extras, Sheet Metal Design.*
3. A dialog box then opens. In this dialog box, define the parameters required for your sheet metal design.
4. If required, you can now select rules that can notify you of possible errors during design.
5. Then begin with the generation of your sheet metal design by drawing closed 2D contours on one another on different workplanes.
6. Define individual plates from the contours.
7. Then join the plates to a set of joins.

8. If required, simulate the bending on a bending machine displayed in simple form.
9. Using the optional module *CADdy++ Sheet Metal Bender* you can finally generate the 2D developed view to your sheet metal design.

3.2 Design

2D contour generation

Sheet metal designs in *CADdy++* are generated from 2D contours using extrusion solids. 2D contours can only be generated on existing workplanes (WP). The *CADdy++ Sheet Metal Design* commands described in the following enable you to string together workplanes for the creation of a sheet metal design.

Firstly begin on an existing WP by generating a closed 2D contour for the first plate. Based on this contour, generate a new workplane, one after the other, for each further side of the plate. Existing workplanes can also be used for further 2D contours.

If all objects on a workplane clearly define the contour of the sheet plate, the workplane is automatically deleted once the solid has been generated using the sheet metal design.

generate WP

Once the command has been started, identify a line or a construction line. As is the case for the *Define WP/WP Set*  command, an individual WP is generated on this line. The line determines the X-axis of the WP. Quit the command with the right mouse button or determine the starting point for a rotation angle at any position using the left mouse button. In this case, the X-axis acts as the rotation axis. The rotation angle can be defined using the cursor or by making an entry in the Status dialog box. Repeat this procedure for each further sheet plate. If you have set ***Parameters / lines in current WP: with copy***, the line is also copied in the new WP and can be used for the new plate.

generate contour by WP intersection

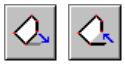
From all the workplanes that are later identified, 2D objects of type *line* are projected onto the WP which was identified first. The individual projection workplanes can be collected with the help of the Ctrl key or by dragging a rectangle. Projections from WP sets can be defined by identifying a WP whilst keeping the Shift key pressed.

generate all contours by WP intersection

If you have simply suggested your sheet metal design using workplanes, this command can be used to generate the required 2D contours on all workplanes in just one step. A number of workplanes can be selected with this command by using the collect command by the Ctrl key, by dragging a rectangle keeping the mouse button pressed or by selecting one or more WP sets keeping the Shift key pressed. Lines of the other WPs involved are projected onto each of the collected workplanes.

edit plate contour

After starting the command, identify one sheet side of your sheet metal design. The solid is displayed as transparent, the plate contour of the 2D contour is visible and can be edited using the 2D commands. When editing the contour, please take note that this remains closed. Modifications are updated using the *Apply* command button in the *Feature Information* dialog box or using the *Update Feature Model*  command on the top 3D toolbar.



show/hide plate contour

The ***show plate contour*** command can be used to open the respective workplane and the contour it contains by identifying one of the sheet sides. As long as you do not make any modifications to the contour and/or no new objects are generated on the WP, the workplane can then be closed using the ***hide plate contour*** command. To do so, identify the frame of the workplane.
If however you have generated new objects on the opened workplane, these are kept and only the original contour elements are hidden once you have used the ***hide plate contour*** command on the WP.

Define and edit set of joins

Once you have drawn closed 2D contours on workplanes, 3D plates can be defined from the contours. All solids of a set of joins result in a complex sheet metal design.

define

Various contours can be collected interactively and extruded from different workplanes. The parameters necessary for this *sheet thickness, material* and *expansion direction* are taken from the current parameter set. The expansion direction is indicated by an arrow on the respective plate and can be modified using the open

dialog box.
The following belong to a set of joins:

- a sheet data Feature

- a set of joins Feature

- one or more plate Features

- a rule Feature for the set of joins

- one rule Feature for each plate Feature

A plate Feature is defined for each contour. All plate Features take their relevant sheet data from the sheet data Feature. Each contour contains an expansion parameter that can be modified independently of the expansion parameters of the plate Features using the 3D Feature Manager.

The set of joins Feature is the managing unit of the plate Features that belong to the set of joins. The contour solid belonging to the set of joins is displayed as transparent and only receives its final representation once the command is quit.

If an automatic rule entry is selected (option card *Parameters* / *Options*), a rule Feature is defined for each set of joins Feature and each plate Feature of the set of joins, in which the available rules and manufacturing profiles are entered.

perform:
Using a window, select bordering 2D line objects. Define the decisive face, by positioning the cursor in a completely bordered section and confirming with the left mouse button.
If you keep the Ctrl key pressed whilst doing this, a number of faces can be collected.
An arrow on the centre of gravity of the solid indicates the expansion direction. In the open dialog box, you can ***Reverse Direction*** if required. After confirming ***Accept Direction***, the solid is generated.
Without quitting the command, a further WP can be activated in order to repeat the procedure.
All solids that you generate during a procedure belong to a **set of joins**.

add plate
The ***add plate*** command can be used to add further plates to an existing set of joins. If, for example, the command is aborted whilst ***join***ing your set of joins, this command can be used to continue the definition of the set of joins.

After starting the command, identify an existing plate. Then select bordering lines of a contour and identify the section within the contour using the left mouse button. If you keep the Ctrl key pressed whilst doing this, a number of faces can be collected.

erase plate

Individual contour solids can be removed from a set of joins by identifying them in the graphics. The respective plate Feature is erased along with the rule Feature, if it only contains this one contour, otherwise only the contour from the plate Feature is removed. If connections exist to this contour, these are erased. Erasing the last plate Feature also leads to the set of joins Feature and sheet data Feature being erased as well as the rule Features connected with it.

add joined plates

This command can be used to unite a separate set of joins to another set of joins. One after the other, identify 2 solids of the respective sets of joins that you wish to become joined together.

3.3 Join

Define, change and erase connections between the contours

define

A connection between two contour solids of a set of joins is defined by identifying one face of each of the solids. If both solids belong to different plate Features of a set of joins, both of these plate Features are united as one. This connection is entered in the respective list of plate Features. If each plate Feature contains one rule Feature each, these are also united as one.
Firstly, the standard connection type of "bending" is entered. The necessary bending radii (inner/outer) and the geometry generating parameters are taken from the current set of parameters (sheet data Feature) and entered in the connection list. In this way, all parameters can be manipulated independently of the sheet data Feature. The bending with the necessary geometry modifications is executed once the connection is entered.

erase

A connection between 2 contours can be selected by picking

clearly. If it is not clear with your first selection, you are requested to select again in order to make it entirely certain what is being chosen. The selected connection is erased from the list. If this results in two separate plates, the plate Feature that contains this connection is also disassembled into 2 plate Features taking the rule Features into account when doing so.

 change

A connection between 2 plates can be edited by identifying one or both solids one after the other. The *Information Feature* dialog box is opened. In this dialog box, amongst other things, you can select the definition of the connection type (bending/welding).

Define and erase rule Features

Rule Features are defined in order to control the verification of the current model according to design and manufacturing conditions. Exactly one rule Feature can be defined for each plate Feature and set of joins Feature. Rule Features can be defined automatically (please refer to *Define set of joins*) or using the following commands.

 **define**

In the list, determine for which plate a rule Feature is to be generated. Once you have selected ***Rule for part*** or ***Rule for group***, identify the respective geometry in the drawing. The rule Feature is defined in the 3D Feature Manager.

 **erase**

In the list, determine for which plate a rule Feature is to be erased. Once you have selected ***Rule for part*** or ***Rule for group***, identify the respective geometry in the drawing. The rule Feature is erased from the 3D Feature Manager.

 **refresh**

Once you have selected a manufacturing profile under ***Parameters/Sheet properties***, the ***refresh*** command can be used to apply the rules of the modified finisher on the current sheet metal design. In this way, the sheet metal design can be recalculated for possible errors according to the modified manufacturing data and the currently determined rules.

perform

This activates or deactivates automatic rule execution during sheet metal design.

Error Report

The "Error Report" command button can be used to display a list with errors of the last rule verification, only if errors have occurred according to the defined rule verifications.

4 Bending Simulation

4.1 Define / erase a developed compound

Bending simulation is carried out separately for each plate. For this, a simple developed view is defined in which the 2D contours are all placed in one plane. Each of these contours is extruded with the defined sheet thickness into a 3D solid. The modelling of these 3D solids is idealised, since the contours are combined to one developed compound without taking into account the bending zones, indentations, stamps or similar.

define

With this command, you can decide whether the bending is to be simulated with or without bending machine.
Firstly, it checks if one or more bending machine(s) have already been loaded in the model.
If a model has already been loaded, the current machine can be identified with the cursor. If the simulation is to take place without a machine, cancel this step with the right mouse button.
If no machine has been loaded, you will be requested to load a bending machine using a file dialog box (as SAT file). If you do not wish to load a machine, quit the dialog box with the **Cancel** command button.
After identifying or loading a bending machine and also after cancelling one of these actions, you can select the plate to be simulated. The developed compound is generated with the respective Features from the chosen plate.
If a bending machine exists, the developed compound is positioned automatically on this. If none exists, the developed compound can be positioned wherever.

erase

Erases the identified developed view geometry and the respective Features from the model.

4.2 Bending

define order of sequence

The basic logic function is the identification of a bending zone in the developed compound. With this, the order of sequence is manipulated in the bending simulation Feature within the bending list.

Further commands can be selected as required from the displayed pop-up menu.

Rotate:

Rotates the developed compound on the bending machine by 180° in regards to the bending edge.

Move:

The developed compound can be moved freely on the bending machine along the bending edge.

Bend:

The current bending zone in the developed compound is arched or smoothed. (For bendings with and without machines.)
If a top tool exists for this bending, it is lowered or lifted depending on the status.

top tool:

A top tool (SAT file) can be selected and inserted for the current bending from a file dialog box. When inserting, the entire length of the tool can be defined in the Status dialog box. Positioning is carried out automatically on the current machine. Movement along the bending edge is possible. An existing tool can be covered.
A tool can be loaded without bending machine, but this is erased after being set down.

bend in order of sequence

The developed compound is bent one after the other according to the order of sequence you have determined or according to the

order of sequence taken over as the default standard.
Firstly, a solid of the compound must be identified.
Loading the command again or picking a solid of the compound can disassemble each further step.

 **add step**

If you have not yet completely defined the bending order of sequence, further bending order of sequences can be determined by identifying the developed compound respectively on each of the bending edges. Otherwise the same command as **define order of sequence** 

 **change order of sequence**

This can be used to change the order of sequence of 2 bendings by picking the respective bending zones. Otherwise the same command as **define order of sequence**  .

Bending zone / verification

generate names

This is used to switch on/off the numbering of bending zones in the current order of sequence.

verification

This is used to switch on/off the verification during bending. The following are verified: collisions of developed compounds with the machine, the top tool and overlapping.

error report

The results of the current verification are listed if errors have been determined according to the defined rule supervision.

Bending machine

 **insert bender geometry**

Via the file selection dialog box you can load a bending machine that has already been defined (as ACIS SAT file) and position it freely within your model.

 **delete bender geometry**

By identifying the machine geometry, this and all Features connected to this are removed from the model.

Store 3D geometry as...

bending machine

This command can be used to define a previously generated 3D geometry as a bending machine and then save it as a SAT file.
Firstly define the faces of the support matrix for the sheet (supporting surface and inner faces of the key) by identifying with the cursor. These faces are tagged with attributes.
Then, all the solids that belong to the machine are selected.
Questions regarding the lower and top tool then appear which can be aborted. Afterwards, the SAT file can be saved in a file dialog box.
The prepared geometry of the bending machine makes it easier to position the developed compounds and the top tools in the machine.

top tool

This command can be used to define a previously defined 3D geometry as a top tool and save it as a SAT file.
Both planar faces of the key and a corner of one of these two faces must be picked. The picked objects are tagged with attributes and simplify positioning when loading in the bending machine. Then, the SAT file is saved using a file dialog box.

5 Rules

5.1 General

This option card can be used to select rules, which are to supervise your sheet metal design. The rules are defined using the knowledge base.

Independent of each other, the supervision can be defined according to the predefined rules for the ***bending plate***, the ***set of joins*** and/or the ***bending simulation***.

Firstly, select the set of rules and allocate them to the section(s) of the sheet metal design to be verified using the  command button. The  command button removes the tagged set of rules from the verification.

6 Restrictions for Working with Sheet Metal Design

6.1 General notes

Contour lines on a bending edge

On bending edges, the 2D contour lines must be congruent for the length of the bend, otherwise no bend is generated.

Edges on bending edge

The endpoint of edges that are to splice together on a bending edge, must lie exactly on each other and on the bending edge or be of a "distinct" distance from each other.

Contours in bending zones

Inner contours which are to lie entirely in the bending zone, must then be introduced to the complete sheet metal design.

Contours that project into the bending zone could lead to problems.

Non-linear edges that splice on a bending edge could lead to problems.

Indentations on bending zones

Generally, indentations are generated on bending zones.

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