

CAD Fixture Recipes

Tiny Inventor, Fusion, and SolidWorks files, each changing one thing at a time, for clean-room native import testing in cadmpeg and SolidCraft. About 15–45 minutes depending on how far you get. Every file helps; partial sets are welcome.

Inventor 2022+ (2025+ ideal)

Fusion (any current build)

SolidWorks (any year; note which)

units: mm

dedicated CC0

How to make them

By hand, from blank

Start every part from the default metric (mm) template. No plugins, scripts, iLogic, or AI driving the app.

One change per twin

Make a, save. Every twin starts from a: open a, change only what that twin says, *Save As* b. Open a again for c, and so on (not a chain $a \rightarrow b \rightarrow c$).

Exact names

Use the IDs below as file names:
P01a.ipt, S03c.ipt, A02b.iam, F09b.f3d,
P01a.sldprt.

Constrain sketches

Fully constrain every sketch, including the point a hole or feature sits on: a coincident to a projected point when there is one, otherwise dimensions from the origin or edges. Don't just drop a point at the spot.

Use the named tool

The method matters as much as the result: the history records *how* you built it. Use exactly the tool and steps given. If you have to do it differently, write what you did on the card.

Your own work

No downloaded, Content Center, or library parts unless the recipe asks for one.

Coordinates. The P01/F01 block spans x −10...10, y −15...15, z 0...10 (20 wide in X, 30 deep in Y, 10 tall). Edges and faces below are named by these coordinates so

every volunteer picks the same one. The sheet in S01/F07 spans x 0...100, y 0...60 on XY.

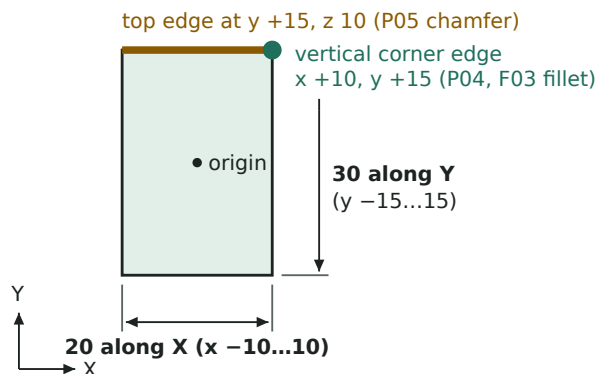


Fig. 1 · P01 / F01 block, seen from above (+Z). 20 wide in X, 30 deep in Y, 10 tall. If your block is 30 along X, it is turned 90°.

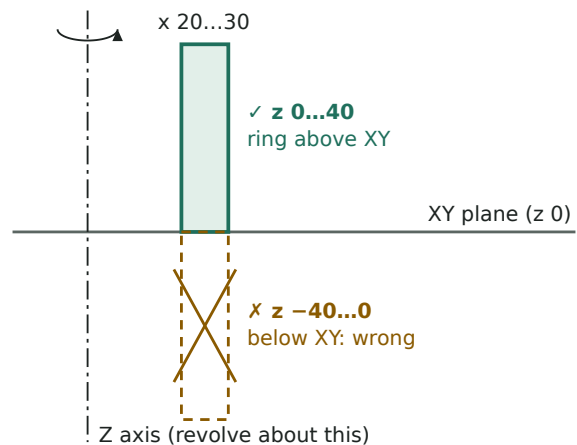


Fig. 2 · P02 / F06 ring profile, on XZ. The sketch's "up" on XZ can point to -Z, so look before you revolve: the profile must sit above the XY plane.

Distinct values matter. Sheet-metal recipes use thickness 2, bend radius 1, and corner gap 0.5 on purpose: when values are equal, the bytes can't show which setting is which.

Inventor

Use one folder for everything and the default project. For assemblies, place the part files from the same folder. Plain *Save / Save As*, not *Save Copy As*.

Parts

ID	MAKE	TWINS
☐ P01	Sketch on XY: rectangle 20 (X) × 30 (Y) centred on the origin, dimensioned. Extrude 10 upward (+Z). Fig. 1 How: Start 2D Sketch on XY. Center Point Rectangle, centre clicked on the origin point. Dimension 20 and 30. Finish Sketch. Extrude, distance 10, one direction.	b: extrude 12

ID	MAKE	TWINS
☐	P02 Sketch on XZ: rectangle from x=20 to x=30, z=0 to z=40, so the ring sits above the XY plane. Revolve 360° about Z. Fig. 2 How: Start 2D Sketch on XZ. Two Point Rectangle, then dimension the corners to x 20/30, z 0/40. The XZ sketch's vertical direction can point to -Z: check the rectangle is above the XY plane (z 0 to 40) before revolving. Revolve about the Z axis.	b: angle 270°
☐	P03 Copy of P01a. Simple hole Ø5 through all, at the centre of the top face (0, 0, 10). How: Hole command. Placement: On Point, using a sketch point at the top-face centre (new sketch on that face, Point at the projected centre). Through All.	b: counterbore Ø9, depth 3
☐	P04 Copy of P01a. Fillet the vertical edge at x = +10, y = +15, R2. Fig. 1 How: Fillet command, Constant tab, select one edge.	b: R3 · c: add the vertical edge at x = -10, y = +15 to the same fillet
☐	P05 Copy of P01a. Chamfer the top edge along X at y = +15, z = 10, distance 1. Fig. 1 How: Chamfer command, Distance mode, select one edge.	b: 1.5
☐	P06 Copy of P01a. Shell, remove the top face (z = 10), thickness 1.5, inside. How: Shell command, select the top face to remove.	b: 2
☐	P07 Plate x 0...60, y 0...40, z 0...5. One Ø4 hole at (5, 5). Rectangular pattern 3 along +X × 2 along +Y, spacing 10. How: Hole command for the first hole, then Rectangular Pattern (Pattern Features) of that hole.	b: 4 × 2
☐	P08 Disc Ø60 × 5 centred on the origin. One Ø6 hole at (20, 0). Circular pattern of 6 about Z. How: Extrude the disc, Hole command for one hole, then Circular Pattern (Pattern Features) about the Z axis.	b: 8

ID	MAKE	TWINS
☐	P09 Copy of P01a. Boss 5×5×5 centred on the +Y end face ($y = +15$), sticking out in +Y. Mirror it across the XZ plane ($y = 0$). How: Sketch on the end face, Extrude Join for the boss, then Mirror (features) with the XZ plane.	b: boss 6 (along X) × 5 × 5
☐	P10 Path on XY: line (0,0) → (40,0) → (40,30), R10 sketch fillet at (40,0). Circle Ø4 on a plane normal to its start. Sweep. How: 2D sketch path with Line + Fillet in the sketch. Work plane Normal to Curve at Point for the circle sketch. Sweep, Path mode.	b: circle Ø5
☐	P11 Square 20 × 20 centred on the origin on XY. Circle Ø10 centred on the Z axis on a work plane offset +30. Loft. How: Work Plane offset from XY. Two sketches. Loft between the two profiles.	b: offset 40
☐	P12 User parameter plate_w = 20 mm. Rectangle plate_w × 30, extrude plate_w/2. How: Parameters dialog, Add Numeric (user parameter). Type the names into the sketch dimension and extrude distance.	b: plate_w = 25
☐	P13 Work plane offset +15 from XY. Circle Ø10 centred on the Z axis, extrude 5 upward. How: Work Plane, Offset from Plane. Sketch on it, Extrude.	b: offset 20
☐	P14 3D sketch: polyline (0,0,0) → (20,0,0) → (20,20,10). Sweep a Ø3 circle along it. How: Start 3D Sketch, Line through the three points (type coordinates). Sweep with a 2D circle on a plane normal to the start.	b: last point z = 15

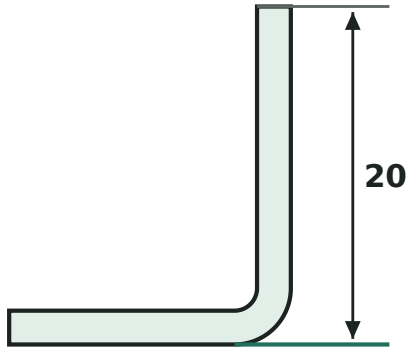
ID	MAKE	TWINS
☐	P15 Copy of P01a. Set the part appearance to the built-in red. How: Appearance override on the part (Appearance dropdown), then for b: select the face and set its appearance.	b: also make the top face blue
☐	P16 Pin: cylinder Ø5 × 20 (for the assembly recipes). How: Sketch on XY, circle, Extrude.	none

Sheet metal

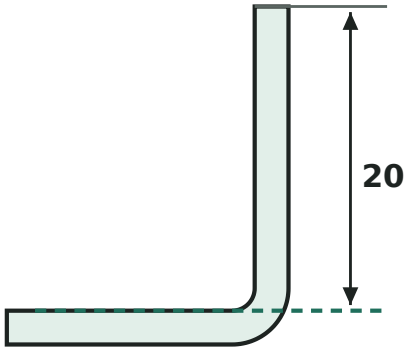
Sheet Metal Defaults: thickness **2**, bend radius **1**, K-factor 0.44, corner relief round, relief size 4, corner gap **0.5**. Set these in **every** S file, and check the bend radius: a new rule may set it to "Thickness" (2), not 1.

ID	MAKE	TWINS
☐	S01 Face from the rectangle x 0...100, y 0...60 on XY, thickness upward. How: New part from the Sheet Metal (mm) template. Sheet Metal Defaults → edit the rule to the values above. Sketch the rectangle, Face command.	b: thickness 2.5
☐	S02 S01a + Flange on the top edge along X at y = 60, height 20, angle 90°, flange going up. Fig. 3 , 4 How: Flange command, Edge select mode, pick the one edge. Distance 20 and angle 90; height datum and bend position left at defaults for a.	b: height 25 · c: angle 60° · d: change only the height datum, i.e. which face the 20 is measured from (Height Extents → Measurement; Fig. 3), keep 20 and 90°; note which option · e: change only the bend position (Fig. 4); note which option

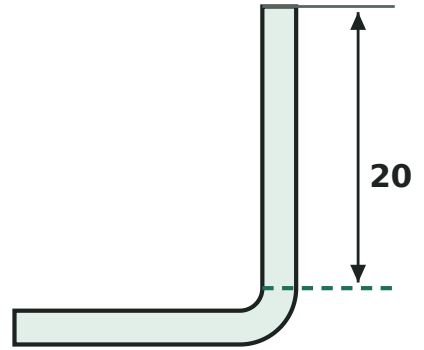
ID	MAKE	TWINS
☐	S03 S01a + one Flange feature on the top edges at $y = 60$ and at $x = 100$, height 20, bend radius 1, meeting at the corner (100, 60). Fig. 5 How: One Flange command with both edges picked (not two separate flanges). Corner tab: relief shape and size per the twins.	b: relief shape square · c: tear · d: relief size 5 · e: corner gap 0.8
☐	S04 S02a + single Hem on the flange's free (top) edge, folded toward $-Y$, gap 1. How: Hem command, Single type, on the flange top edge.	b: gap 1.5
☐	S05 S03a + Create Flat Pattern, then Go To Folded Part, save. How: Create Flat Pattern from the ribbon, then Go to Folded Part, then save.	none
☐	S06 S02a with bend relief shape round, relief width 3. How: Flange command, Bend tab: relief shape and width.	b: straight
☐	S07 S01a + Corner Round R3 on the corner at (100, 60). How: Corner Round command (sheet metal), pick one corner of the face.	b: R5

A

from the outside face
(the sheet's bottom, z 0)

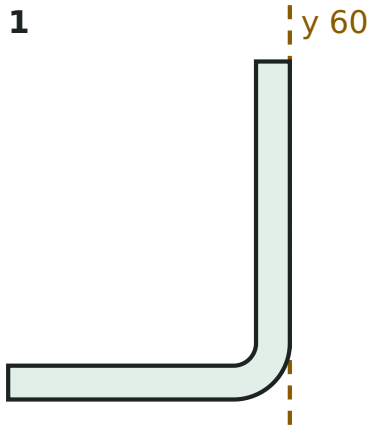
B

from the inside face
(the sheet's top, z 2)

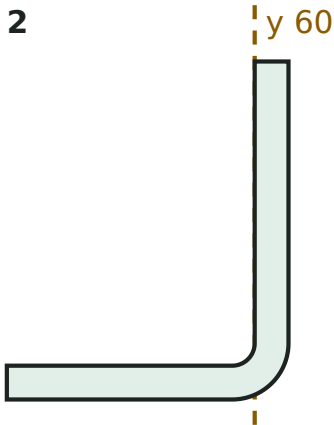
C

from where the bend ends
(tangent to the bend)

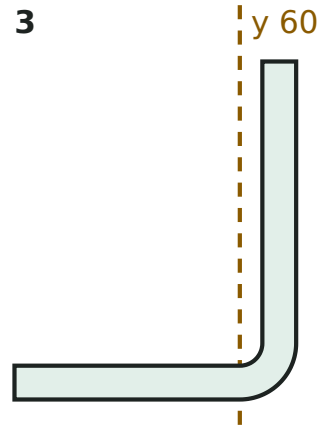
Fig. 3 · S02d / F08: the height datum. Side view along X; the flange stands at $y = 60$, thickness and bend drawn larger than scale. The height stays 20 in every case; only the face it is measured from changes. S02a: leave the default. S02d: pick a different one and write which (A, B, or C, or the app's name for it) on the card.

1

outside face of the flange
on the edge (flange inside it)

2

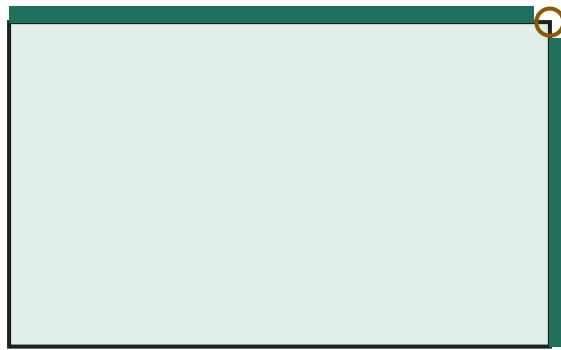
inside face of the flange
on the edge (flange outside it)

3

the bend starts at the edge
(the bend is added outside)

Fig. 4 · S02e: the bend position. The dashed line is the edge of the S01 face ($y = 60$) before the flange. The options move the flange relative to that edge. S02e: change only this, keep height 20, 90° , and the default height datum, and note which option.

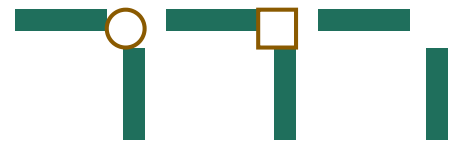
flange on y 60, height 20, up



corner (100, 60)

flange on x 100
height 20, up

corner close-ups:
relief size 4 (d: 5)



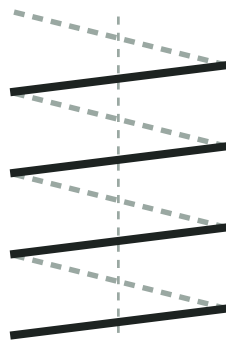
a: round b: square c: tear

gap between flanges
0.5 (e: 0.8)

.00 along X · 60 along Y · one Flange feature

Fig. 5 · S03 corner, seen from above. One Flange feature with both edges picked, bend radius 1. The twins change only the corner: the relief shape, its size, or the gap. Each change goes in that flange's corner settings; check it shows before saving.

a: right-hand

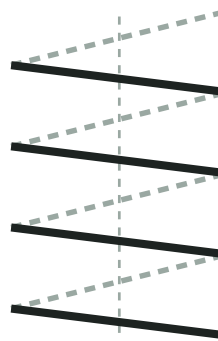


seen from the side



from above: counter-
clockwise as it climbs

b: left-hand



seen from the side



from above: clockwise
as it climbs

Fig. 6 · F13 coil hands. Solid lines are the front of the wire, dashed the back. a, right-hand (like a standard screw): looking down from above (+Z), the wire turns counter-clockwise as it climbs. b, left-hand: looking down from above, it turns clockwise as it climbs. In Fusion, a is the rotation checkbox as it comes (unticked) and b is the box ticked.

Assemblies

Place P01a first, then P02a (and P03a / P16 where named). Inventor grounds the first part by default; leave it unless the twin says otherwise.

ID	MAKE	TWINS
☐	A01 Place P01a and P02a, no constraints. How: New assembly from the standard (mm) template. Place Component P01a (it is grounded automatically), then place P02a off to the side.	b: unground P01a · c: ground both · d: hide P02a (visibility off) · e: suppress P02a
☐	A02 A01a + one Mate: P02a bottom face (z = 0) to P01a top face (z = 10), offset 0. How: Constrain command, Mate type, face to face.	b: offset 5
☐	A03 A01a + one Flush: P02a bottom face to P01a bottom face, offset 0. How: Constrain command, Mate type with Flush solution.	b: offset 5
☐	A04 A01a + one Angle constraint between P01a top face and P02a top face, 30°. How: Constrain command, Angle type, Directed Angle, two faces.	b: 45°
☐	A05 P03a + P16. Insert: pin bottom circular edge into the hole top edge. How: Constrain command, Insert type, hole edge to pin end edge.	b: offset 2
☐	A06 P01a + P16. Tangent: the pin's cylindrical face to P01a top face (z = 10). How: Constrain command, Tangent type, cylinder face to planar face.	b: outside ↔ inside
☐	A07 A01a + rectangular component pattern of P02a, 3 × 1, spacing 70. How: Pattern Component, Rectangular tab.	b: 4 × 1
☐	A08 A01a + a second model state where P02a is suppressed. Save with the master active. How: Model browser, Model States folder, New. In the new state, suppress P02a. Activate the master again before saving.	b: save with the second state active

ID	MAKE	TWINS
☐	A09 A01a + a View Representation "View1" where P02a is hidden. Save with Default active. How: Representations folder, View, New View Representation, rename View1, turn off P02a visibility. Activate Default before saving.	b: save with View1 active
☐	A10 A01a + colour override in the assembly: P01a blue. How: Select P01a in the assembly and set its appearance override from the Appearance dropdown.	b: green

Fusion

New design for each part, units mm. Export each with *File* → *Export* → *.f3d*; assemblies also as *.f3z*.

ID	MAKE	TWINS
☐	F01 Sketch on XY: rectangle 20 (X) × 30 (Y) centred on the origin, dimensioned. Extrude 10 upward (+Z). How: Create Sketch on XY. Center Rectangle tool, centre clicked on the origin. Sketch Dimension 20 and 30: the 30 runs along Y (vertical in the sketch), as in Fig. 1. Finish Sketch. Extrude, One Side, 10, New Body.	b: 12
☐	F02 User parameter plate_w = 20 mm; rectangle plate_w × 30, extrude plate_w/2. How: Modify → Change Parameters, add a User Parameter. Type the names into the dimension and extrude distance.	b: plate_w = 25

ID	MAKE	TWINS
☐	F03 F01a + fillet the vertical edge at $x = +10$, $y = +15$, R2 (the corner edge, not a top edge; Fig. 1). How: Fillet tool, select one edge, Constant radius.	b: R3
☐	F04 F01a + hole $\varnothing 5$ through, at the top face centre (0, 0, 10). How: Hole tool (not an extrude cut), placed with a point on the top face at its centre, Simple, Through All.	b: counterbore $\varnothing 9$ depth 3
☐	F05 Plate $x\ 0\ldots 60$, $y\ 0\ldots 40$, $z\ 0\ldots 5$ + $\varnothing 4$ hole at (5, 5), rectangular pattern 3 along +X $\times$ 2 along +Y, spacing 10. How: Hole tool for one hole, then Rectangular Pattern, type Features, on that hole.	b: 4×2
☐	F06 Revolve: rectangle $x\ 20\text{--}30$, $z\ 0\text{--}40$ (above the XY plane), about Z, 360° . Fig. 2 How: Create Sketch on XZ, Two-Point Rectangle, dimension. Check the rectangle sits above the XY plane ($z\ 0$ to 40), not below. Revolve about the Z axis.	b: 270°
☐	F07 Sheet metal rule: thickness 2, bend radius 1, gap 0.5. Flange from the rectangle $x\ 0\ldots 100$, $y\ 0\ldots 60$. How: Sheet Metal tab. Sheet Metal Rules, new rule with these values. Sketch the rectangle, Flange from the profile.	b: thickness 2.5

ID	MAKE	TWINS
☐	F08 F07a + edge flange on the top edge at y = 60, height 20, going up. Fig. 3, 4 How: Flange tool on the edge (Edge selection), Height datum and bend position left at defaults unless the twin says.	b: height 25 · c: from a (height 20), edges at y = 60 and x = 100, meeting at (100, 60)
☐	F09 F08c + Create Flat Pattern, finish, save. How: Create Flat Pattern, Finish Flat Pattern, save.	none
☐	F10 High priority Two components: the F01a block and a pin Ø5 × 20; block grounded. Rigid joint: pin bottom centre to block top-face centre (0, 0, 10). How: New Component for each, Ground the first. Joint tool (not As-Built), snapping to faces.	b: revolute · c: slider · d: unground · e: hide the pin · f: from c, the slider: drive the pin 5 mm up (right-click the joint, Drive Joints, distance 5, OK), so its bottom is at z = 15; don't Capture Position · g: from b, the revolute: drive it 90° (Drive Joints, angle 90, OK); don't Capture Position · h: the rigid joint with an offset and an angle: right-click the joint, Edit Joint, Offset Z 5 mm and Angle 30°, OK. The pin rises 5 mm (its bottom at z = 15) and turns 30° about the joint's Z; the pin looks the same turned, so check Edit Joint shows 30°
☐	F11 F01a with the body appearance set to a red paint. How: Appearance dialog, drag the paint onto the body; for b, onto the top face.	b: top face blue

ID	MAKE	TWINS
☐	F12 High priority Copy of F10a, saved with no change. Each twin changes only the joint's motion type and axis, or its limits. Keep the two snap points; don't move or animate the pin. How: For each twin, open F12a, right-click the joint under Joints in the browser, Edit Joint, set the motion type and axis the twin names, OK. For limits, right-click the joint, Edit Joint Limits, tick only the limits the twin names, type the values, OK. The axes are the joint's own: its Z points up, out of the block's top face. If Fusion offers other axis names, pick the X, Y, or Z named and note what you saw.	b: revolute, rotate about Z · c: revolute, rotate about X · d: slider, slide along Z · e: slider, slide along X · f: cylindrical, about Z · g: pin-slot, rotate about Z, slide along X · h: planar, normal Z · i: ball, the default axes (pitch Z, yaw X) · j: revolute about Z, rotation limits minimum -30°, maximum 45°, rest 10° · k: slider along Z, slide limits minimum -2, maximum 5, rest 1 mm · l: revolute about Z, rotation minimum -30° only (maximum off)

ID	MAKE	TWINS
☐	F13 High priority A right-hand coil, like a standard screw thread: looking down from above (+Z), the wire turns counter-clockwise as it climbs. About the Z axis, centred on the origin, starting on XY: diameter 20, pitch 5, 4 revolutions, angle 0, round wire 2 mm thick (Section Size 2) on the centre line. Fig. 6 How: Create → Coil, New Body. Pick the XY plane, click the origin for the centre, type 20 for the diameter. Type: Revolution and Pitch; Revolutions 4, Pitch 5, Angle 0. Section: Circular, Section Position On Center, Section Size 2 (the wire's diameter; Fusion's default is thicker than the pitch, and the turns would collide). Leave Fusion's rotation checkbox as it comes (unticked): that makes the right-hand coil. Check it: looking down from above, the wire turns counter-clockwise as it climbs.	b: the left-hand coil, its mirror image: looking down from above (+Z), the wire turns clockwise as it climbs. Tick Fusion's rotation checkbox and change nothing else (Fig. 6)

ID	MAKE	TWINS
☐ F14	High priority A Box primitive (not a sketch and extrude): first corner at the origin, 20 along +X, 30 along +Y, 10 up, so it spans x 0...20, y 0...30, z 0...10. How: Create → Box, New Body. Pick the XY plane, click the origin for the first corner, then type Length 20 and Width 30, dragging toward +X and +Y, and Height 10 upward. Check the extents above before saving.	b: new design: the same box with its first corner on a sketch point at (5, −7) (sketch on XY, Point, dimension it 5 and 7 from the origin, Finish Sketch, then Box with the first corner snapped to that point), so it spans x 5...25, y −7...23, z 0...10 · c: new design: the same box on the XZ plane, first corner at the origin, 20 along +X, 30 along +Z, height 10 off the plane; note whether it went to +Y or −Y · d: new design: a Cylinder primitive (Create → Cylinder): an offset plane 3 above XY, a sketch point on it at (5, −7), the cylinder's centre snapped to that point, diameter 8, height 12 upward, so it spans x 1...9, y −11...−3, z 3...15

SolidWorks

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs) with the SolidWorks tools below, and save them as P01a.sldprt, S02b.sldprt, A02a.sldasm. Start from the default metric (MMGS) part and assembly templates. Same rules: one change per twin, every twin from a, fully constrained sketches. If a step has no exact match, use the closest tool and write what you did on the card.

RECIPE SAYS	IN SOLIDWORKS
Planes XY · XZ · YZ (coordinates as written)	Front · Top · Right planes. SolidWorks shows Y up; keep the recipe's x, y, z values as they are, so "upward (+Z)" means toward the viewer from the Front plane
Extrude · Revolve · Sweep · Loft	Extruded Boss/Base · Revolved Boss/Base · Swept Boss/Base · Lofted Boss/Base
Hole command (P03, P07, P08)	Hole Wizard, Hole type (b: Counterbore), position point on the face, fully dimensioned

RECIPE SAYS	IN SOLIDWORKS
Fillet · Chamfer · Shell	Fillet (Constant size) · Chamfer (Distance) · Shell
Rectangular / Circular Pattern (features) · Mirror	Linear Pattern · Circular Pattern (Features and Faces) · Mirror (feature, about the plane the recipe names, mapped as in the first row)
Work plane offset · normal to curve	Reference Geometry → Plane: Offset · Normal to curve at a point
User parameter plate_w (P12)	Equations → Global Variable "plate_w" = 20, used in the dimension and the extrude depth
3D sketch (P14)	3D Sketch, Line through the three points
Appearance (P15, A10)	Appearances: part red; b: top face blue · in the assembly, component appearance blue
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	Base Flange/Tab sheet-metal parameters: thickness 2, bend radius 1, K-Factor 0.44, auto relief Round, relief ratio or size as close to 4 as allowed; note what you set
Face (S01) · Flange (S02, S03) · Hem (S04)	Base Flange/Tab · Edge Flange (S03: both edges in one Edge Flange) · Hem (Closed/Open, gap 1)
Height datum · bend position (S02d/e)	Edge Flange → Flange Length reference (Outer/Inner virtual sharp, Tangent) · Flange Position (Material Inside/Outside, Bend Outside)
Corner relief / gap (S03) · Corner Round (S07)	Edge Flange corner relief and gap · Break-Corner/Corner-Trim, Fillet R3
Flat Pattern, back to folded (S05)	Flatten on, then off, save with it off
Ground / unground (A01)	Fix / Float
Mate · Flush · Angle (A02-A04)	Coincident (face to face, b: Distance mate 5) · Coincident, aligned · Angle mate

RECIPE SAYS	IN SOLIDWORKS
Insert (A05) · Tangent (A06)	Concentric + Coincident on the edges (b: Distance 2) · Tangent mate
Pattern Component (A07)	Linear Component Pattern, 3 × 1, spacing 70
Model state (A08) · View representation (A09)	Configurations (second configuration with P02a suppressed) · Display States (P02a hidden)

Send them

Zip the folder and send it with this card filled in. The CC0 line lets the files go into cadmpeg's public test corpus.

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App and exact version (Help → About):

Operating system:

License type (commercial / education / startup / hobby / personal):

Date made:

Files included:

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Every feature is my own work; no downloaded or library parts.

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