Version 1.61 · 2026-11-20

CAD Fixture Recipes

Tiny Inventor, Fusion, SolidWorks, Creo, CATIA, NX, Solid Edge, Alibre Design, and Onshape 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) · Creo (any release; note which) · CATIA (V5 file release; note which) · NX (any listed release; note which) · Solid Edge (any listed year; note which) · Alibre Design (any listed release; note which) · Onshape (current) · 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→b→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.

Revisions. A recipe that has changed since it was first published shows its revision under its ID, like `r2`. Files you made from an earlier revision still count. We ask for one again only when the change needs a new file.

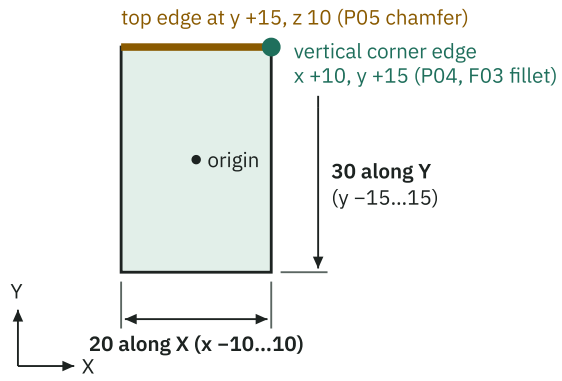


Fig. 1 · P01 / F01 block, seen from above (+Z). 20 wide in X, 30 deep in Y, 10 tall. If your block is 30 along X, it is turned 90°.

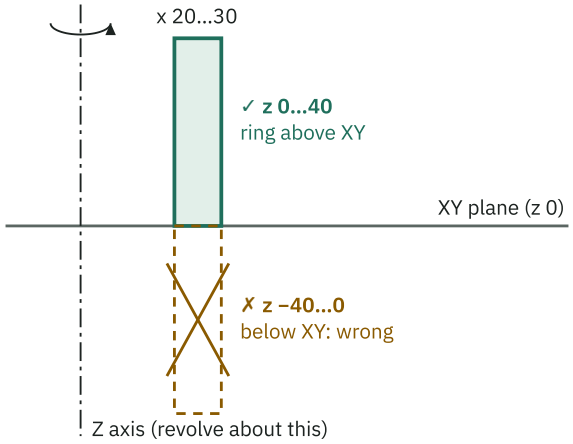


Fig. 2 · P02 / F06 ring profile, on XZ. The sketch's "up" on XZ can point to −Z, so look before you revolve: the profile must sit above the XY plane.

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*. Use your newest Inventor (2025 or 2026 is ideal). If you also have Inventor 2022 or older, make P01a there too: on the portal, add that version as a second app.

Parts

	ID	MAKE	TWINS
☐	**P01** r2	Sketch on XY: rectangle 20 (X) × 30 (Y) centred on the origin, dimensioned. Extrude 10 upward (+Z). Fig. 1 Reference drawing (PNG) 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
☐	**P02** r4	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 Reference drawing (PNG) 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** r2	Copy of P01a. Simple hole Ø5 through all, at the centre of the top face (0, 0, 10). Reference drawing (PNG) 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

	ID	MAKE	TWINS
☐	**P04** r5	Copy of P01a. Fillet the vertical edge at x = +10, y = +15, R2. Fig. 1 Reference drawing (PNG) How: Fillet command, Constant tab, select one edge.	b: R3 · c: add the vertical edge at x = −10, y = +15 to the same fillet · d: edit a's fillet to a variable radius (Variable tab) on the same vertical edge (x = +10, y = +15): R1 at its bottom end (z = 0, on the XY plane) and R3 at its top end (z = 10). Add no radius points between them and change nothing else. Leave Smooth radius transition, or any setting for how the radius changes between the ends, as it comes, and write what it shows on the card · e: from **d**: edit the same variable-radius fillet on the vertical edge at x = +10, y = +15, changing only the two radii: R3 at its bottom end (z = 0, on the XY plane) and R1 at its top end (z = 10). Add no radius points between them and change nothing else. Save As P04e

	ID	MAKE	TWINS
☐	**P05** r4	Copy of P01a. Chamfer the top edge along X at y = +15, z = 10, distance 1. Fig. 1 Reference drawing (PNG) How: Chamfer command, Distance mode, select one edge.	b: 1.5 · c: edit a's chamfer to Two Distances on the same edge (the top edge along X at y = +15, z = 10): Distance 1 = 1, Distance 2 = 2. Leave the sides as Inventor puts them (don't press Flip) and change nothing else. Write on the card which face got the 2: the top face or the +Y end face · d: from **c**: edit the chamfer, press Flip once, so the 1 and the 2 swap faces, OK. Change nothing else, and write on the card which face got the 2 · e: edit a's chamfer to Distance and Angle on the same edge: Distance 2, Angle 30°. If Inventor asks for a face, pick the top face (z = 10). Leave the sides as they come (don't press Flip) and change nothing else. Write on the card which face the 2 runs along: the top face or the +Y end face · f: from **e**: edit the chamfer, press Flip once (or, if there is no Flip, pick the +Y end face instead of the top face), OK, so the 2 runs along the other face. Change nothing else, and write on the card which face the 2 runs along
☐	**P06**	Copy of P01a. Shell, remove the top face (z = 10), thickness 1.5, inside. Reference drawing (PNG) How: Shell command, select the top face to remove.	b: 2

	ID	MAKE	TWINS
☐	**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. Reference drawing (PNG) 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. Reference drawing (PNG) How: Extrude the disc, Hole command for one hole, then Circular Pattern (Pattern Features) about the Z axis.	b: 8
☐	**P09** r3	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). Reference drawing (PNG) 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. Reference drawing (PNG) 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. Reference drawing (PNG) How: Work Plane offset from XY. Two sketches. Loft between the two profiles.	b: offset 40

	ID	MAKE	TWINS
☐	**P12** r2	User parameter `plate_w = 20 mm`. Rectangle plate_w × 30, extrude `plate_w/2`. Reference drawing (PNG) 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. Reference drawing (PNG) How: Work Plane, Offset from Plane. Sketch on it, Extrude.	b: offset 20
☐	**P14** r2	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
☐	**P15**	Copy of P01a. Set the part appearance to the built-in red. Reference drawing (PNG) 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). Reference drawing (PNG) How: Sketch on XY, circle, Extrude.	none

Feature history

Inventor only. Every sketch on XY unless it says otherwise, fully dimensioned from the origin. Leave every name as Inventor gives it (Sketch1, Extrusion1).

	ID	MAKE	TWINS
☐	**X01**	**High priority** Two separate rectangles in one sketch: x 0…20, y 0…10 and x 30…50, y 0…10. One Extrude, distance 5, taking both. How: Start 2D Sketch on XY. Two Point Rectangle twice, dimension each corner from the origin. Finish Sketch. Extrude: click the left rectangle, then the right one, distance 5.	b: the same, but click the right rectangle first, then the left · c: the right rectangle only
☐	**X02**	**High priority** Rectangle 40 × 30 centred on the origin with a circle Ø10 at the origin inside it. Extrude 5, taking only the ring (the rectangle with the hole). How: Center Point Rectangle and Circle, both centred on the origin point, dimensioned. Extrude: click between the circle and the rectangle, distance 5. The disc stays empty: the part has a hole.	b: edit Extrusion1 (right-click, Edit Feature) to take only the inner disc · c: edit Extrusion1 to take both the ring and the disc, so the block is solid
☐	**X03**	**High priority** Two overlapping rectangles: x 0…30, y 0…20 and x 20…50, y 10…30. They make 3 regions; the overlap is the square x 20…30, y 10…20. Extrude 5, taking only the overlap square. How: Two Point Rectangle twice, each corner dimensioned from the origin (don't constrain the rectangles to each other). Extrude: click inside the overlap square only, distance 5.	b: edit Extrusion1 to take the whole first rectangle: the overlap and the L-shaped region next to it · c: edit Extrusion1 to take all 3 regions

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☐	**X04**	Copy of P01a. On the top face (z = 10), a profile closed by the block's edge: lines from (−5, 15) to (−5, 5) to (5, 5) to (5, 15), closed by the top edge at y = +15. Extrude Join 5 upward. How: Start 2D Sketch on the top face. Project Geometry the top edge at y = +15 (if Inventor already projected the face's edges, use that line). Three lines from the projected edge and back to it, dimensioned. Extrude, pick the 10 × 10 region, Join, distance 5 upward.	b: edit Extrusion2 to Cut, distance 3 into the block
☐	**X05**	One sketch used by two extrudes: rectangles x 0…20, y 0…20 and x 30…50, y 0…20. Extrusion1 takes the left one, distance 5; Extrusion2 takes the right one, distance 10. How: Both rectangles in Sketch1, dimensioned. Extrude the left one 5. In the browser, right-click Sketch1 under Extrusion1, Share Sketch. Extrude the right one 10 as a second feature (not an edit of the first).	b: edit Extrusion2 to distance 15
☐	**X06**	High priority A D shape: a line from (−10, 0) to (10, 0), closed by an arc from (10, 0) through (0, 10) to (−10, 0). Extrude 5. How: Line, then Three Point Arc: click (10, 0) first, then (−10, 0), then the top point; dimension it (radius 10, centre on the origin). Extrude 5.	b: the same shape, but draw the arc the other way: click (−10, 0) first, then (10, 0) · c: new design: a slot: lines from (−10, −5) to (10, −5) and from (10, 5) to (−10, 5), with a Tangent Arc closing each end (R5); extrude 5

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☐	**X07** r2	Rectangle 20 × 30 centred on the origin with a construction line from corner (−10, −15) to corner (10, 15). Extrude 10, taking the whole rectangle. How: Center Point Rectangle. Inventor adds both diagonals as construction lines by itself: keep them as they come and don't draw another. The one from (−10, −15) to (10, 15) is the line the twin uses. Extrude 10.	b: edit Sketch1, set the diagonal from (−10, −15) to (10, 15) back to a normal line (the other stays construction), then edit Extrusion1 to take both triangles
☐	**X08**	Copy of P01a. Edit Sketch1: sketch Fillet R3 on the corner at (10, 15). Finish; the extrude updates. How: In the browser, right-click Sketch1 under Extrusion1, Edit Sketch. Fillet (sketch), radius 3, click the two lines meeting at (10, 15). Finish Sketch.	b: R5
☐	**X09**	A closed spline through (−20, 0), (0, −10), (20, 0), and (0, 10). Extrude 5. How: Spline (Interpolation), click the four points, close it on the first point. Dimension the points from the origin. Extrude 5.	none

	ID	MAKE	TWINS
☐	**X10**	**High priority** A coil around the Y axis: on XY, a circle Ø2 centred at (10, 0). Coil with pitch 5 and 4 revolutions, climbing toward +Y. How: Start 2D Sketch on XY. Circle, dimension Ø2 and its centre from the origin. Finish Sketch. Coil: click the circle as the Profile and the Y Axis under Origin in the browser as the Axis. Check that the preview climbs toward +Y; click the direction arrow beside Axis only if it doesn't, and tell us in the note if you had to. Leave Rotation as it comes. On Coil Size, Type Pitch and Revolution, Pitch 5, Revolution 4, Taper 0. Leave Coil Ends as they come. Each twin starts again from the sketch.	b: click the direction arrow once from a, so it climbs toward –Y · c: from a, choose the other Rotation button, so the wire turns the other way · d: Type Revolution and Height, Revolution 4, Height 20: the same shape as a · e: Type Pitch and Height, Pitch 5, Height 20: the same shape as a · f: Type Spiral, Pitch 5, Revolution 4: the wire stays flat and spirals in one plane · g: a tilted axis, direction (0.6, 0.8, 0). In the same sketch, a Line from (0, 0) to (30, 40), and the circle Ø2 centred at (8, –6) instead. Click the line as the Axis. Pitch 5, Revolution 4, climbing toward (30, 40)

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☐	**X11**	High priority A D shape extruded 30, and a square post extruded To its curved face. The D: a line from (−10, 0) to (10, 0), closed by an arc from (10, 0) through (0, 10) to (−10, 0), Extrude 30 upward. The post: on a work plane at y = 30, a rectangle x −3…3, z 12…18, Extrude Join toward −Y, Extents To the D's curved face. How: Start 2D Sketch on XY. The D as in X06: Line, then Three Point Arc, radius 10 with its centre on the origin. Finish Sketch. Extrude 30 upward. Work Plane, Offset from Plane: click the XZ plane under Origin, offset 30. Check it sits at y = +30, on the curved side of the D; if it went to −30, type −30 instead and note it. Start 2D Sketch on that work plane. Two Point Rectangle, dimension its sides from the origin to x −3 and 3 and z 12 and 18 (check the square is above the XY plane). Finish Sketch. Extrude, pick the square, Join, Extents To, then click the curved face of the D. Leave the other options as they come.	b: from a, edit the post's Extrude and change only its To face: click the D's flat face at y = 0 instead of the curved face. The post then runs into the D up to y = 0, so the part looks the same as a: that's fine. If Inventor won't take that face, tell us in the note

	ID	MAKE	TWINS
☐	**X12**	**High priority** A plate x −30…30, y −20…20, extruded 5 up, with one hole Ø4 through at the origin. Rectangular Pattern of the hole: 3 along X, spacing 10, and 2 along Y, spacing 10, so with the default directions the holes sit at x 0, 10, and 20 and y 0 and 10. How: Center Point Rectangle 60 × 40 on the origin, Extrude 5 up. Start 2D Sketch on the top face, Point on the projected origin (if the origin isn't there, Project Geometry the Center Point from the browser). Finish Sketch. Hole: Simple Hole, Drilled, Through All, Ø4, on that point. Rectangular Pattern: Features the hole. Direction 1: click the X Axis under Origin in the browser, Count 3, Spacing 10. Direction 2: click the Y Axis, Count 2, Spacing 10. Don't click either Flip button for a; if the preview runs toward −X or −Y, tell us in the note.	b: from a, edit the pattern and click Flip for Direction 1 only, so the holes run toward −X (x 0, −10, and −20) · c: from a, click Flip for Direction 2 only, so they run toward −Y (y 0 and −10)
☐	**X13**	**High priority** Sketch on XZ: rectangle x 20…30, z 0…40, above the XY plane. Revolve about the Z axis, Extents Angle 270°. How: Start 2D Sketch on XZ, Two Point Rectangle, dimension its corners to x 20 and 30 and z 0 and 40. Check the rectangle is above the XY plane (z 0 to 40). Finish Sketch. Revolve: the rectangle, Axis the Z Axis under Origin, Extents Angle, 270. Leave the direction as it comes.	b: from a, edit the Revolve and change only its direction (the direction buttons beside Angle), still 270°

	ID	MAKE	TWINS
☐	**X14**	**High priority** Copy of P01a. A boss 5 × 5 × 5 centred on the +Y end face (y = +15), sticking out in +Y, as in P09a. Fillet R1 on the boss's four edges that run along Y. Then one Mirror of both the boss's Extrusion and the Fillet across the XZ plane (y = 0). How: Sketch on the end face at y = +15, a 5 × 5 square centred on the face, dimensioned. Extrude Join 5 in +Y. Fillet, Constant, R1, click the four edges of the boss that run out from the end face along Y. Mirror: Features, click the boss's Extrusion and the Fillet in the browser, Mirror Plane the XZ Plane under Origin.	b: from a, edit the Mirror so it takes only the boss's Extrusion, not the Fillet: the mirrored boss keeps sharp edges
☐	**X15**	**High priority** Copy of P01a. Chamfer, Distance 1, on the top edge along Y at x = +10, z = 10. Then, as the last feature, Fillet R2 on the vertical edge at x = +10, y = +15, which the chamfer has already cut short at its top end. How: Open P01a and Save As X15a. Chamfer, Distance, 1, click the one top edge along Y at x = +10 (z = 10). OK. Then Fillet, Constant, R2, click the vertical edge at x = +10, y = +15. OK. The Fillet must be the last feature in the browser.	b: from **a**: delete the Chamfer in the browser (keep the Fillet, on the same vertical edge, still last), then Save As X15b. Change nothing else

Extents and operations

Inventor only. Same rules as above.

	ID	MAKE	TWINS
☐	**E01**	**High priority** Two work planes, offset +12 and −3 from XY. Then a rectangle 20 × 30 centred on the origin on XY, Extrude distance 10 upward. How: Work Plane, Offset from Plane: XY, 12. Again: XY, −3. Then the sketch and the extrude as in P01. Each twin changes only the extents of Extrusion1 (right-click, Edit Feature).	b: Symmetric, distance 10 · c: Asymmetric: 10 upward, 4 downward · d: To: the work plane at 12 · e: Between: the work plane at −3 and the one at 12 · f: distance 10 with Taper −5°, so the block narrows as it rises
☐	**E02**	Copy of P01a. A circle Ø16 at the origin on XY, Extrude 15 upward as a New Solid (a second body). How: Start 2D Sketch on XY, circle at the origin point, dimension 16. Extrude, distance 15, New Solid. Each twin changes only that extrude's operation.	b: Join · c: Cut · d: Intersect · e: Cut, extents Through All
☐	**E03**	**High priority** A block 100 × 60 × 10: rectangle from (0, 0) to (100, 60) on XY, Extrude 10 upward. A work plane offset 30 from the YZ plane (at x = 30). Split, method Split Solid, tool the work plane: two bodies, x 0…30 and x 30…100. How: Two Point Rectangle on XY, corner on the origin point, dimensioned. Extrude 10. Work Plane, Offset from Plane: YZ, 30. Modify panel, Split: Split Solid, Split Tool the work plane, OK. Each twin changes only Split1 (right-click, Edit Feature).	b: Trim Solid, remove the +X side (keep x 0…30) · c: Trim Solid, remove the −X side (keep x 30…100) · d: Split Face, Faces: Select, only the top face (z = 10) · e: before Split1, a surface at x = 30: sketch on XY a line from (30, −10) to (30, 70), Extrude it as a Surface, symmetric, distance 30; then edit Split1 to use that surface as the Split Tool instead of the work plane

Sheet metal

Sheet Metal Defaults: thickness **2**, bend radius **1**, K-factor 0.44, corner relief round, corner relief size 4, corner gap **0.5**. Leave the bend relief (shape, width, and depth) as it comes. 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** r2	Face from the rectangle x 0…100, y 0…60 on XY, thickness upward. Reference drawing (PNG) 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 Reference drawing (PNG)

	ID	MAKE	TWINS
☐	**S02** r6	S01a + Flange on the top edge along X at y = 60, height 20, angle 90°, flange going up. Fig. 3, 4 Reference drawing (PNG) 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 Reference drawing (PNG) · c: angle 60° Reference drawing (PNG) · 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 Reference drawing (PNG) · e: change only the bend position (Fig. 4); note which option Reference drawing (PNG) · f: change only the flange's width: Width Extents type Width, Offset 20 from the corner at x = 0, Width 50, so the flange spans x 20…70 Reference drawing (PNG) · g: change only the flange's width: Width Extents type Offset, Offset 1 is 10 from x = 0 and Offset 2 is 30 from x = 100, so the flange spans x 10…70 Reference drawing (PNG) · h: change only the bend relief depth to 1 (Bend tab, Relief Depth), shape and width left as they come · i: change only the bend transition (Bend tab, Bend Transition) to Arc, bend relief left as it comes. If it already shows Arc, choose Straight Line instead and note which. The folded part may look the same as a: that's fine
☐	**S03** r7	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 Reference drawing (PNG) 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 Reference drawing (PNG) · c: tear · d: relief size 5 Reference drawing (PNG) · e: corner gap 0.8 Reference drawing (PNG) · f: on the Bend tab (not the Corner tab), bend relief shape round, its width as it comes; corner settings untouched

	ID	MAKE	TWINS
☐	**S04** r2	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 · c: type Double, its other values as they come · d: type Teardrop, its other values as they come · e: type Rolled, its other values as they come · f: the same single hem folded toward +Y (flip its direction) · g: the single hem with Length 12
☐	**S05**	S03a + Create Flat Pattern, then Go To Folded Part, save. Reference drawing (PNG) How: Create Flat Pattern from the ribbon, then Go to Folded Part, then save.	none
☐	**S06** r2	S02a with bend relief shape round, relief width 3. Reference drawing (PNG) How: Flange command, Bend tab: relief shape and width.	b: straight Reference drawing (PNG) · c: relief shape back to the default (as S02a has it), width 3 kept · d: relief width back to the default (as S02a has it), shape round kept
☐	**S07** r2	S01a + Corner Round R3 on the corner at (100, 60). Reference drawing (PNG) How: Corner Round command (sheet metal), pick one corner of the face.	b: R5 Reference drawing (PNG) · c: R3 on all four corners of the face, in one Corner Round Reference drawing (PNG)

	ID	MAKE	TWINS
☐	S08	High priority Inventor only A Contour Flange as the first feature (the Contour Flange command, not Face or Flange): an open L on XZ from (0, 0) to (50, 0) to (50, 30), above the XY plane, 40 wide along Y. The result is one bent sheet: a 50 long floor and a 30 tall wall, 40 wide. Reference drawing (PNG) How: New part from the Sheet Metal (mm) template, Sheet Metal Defaults as above. Start 2D Sketch on XZ: two lines, dimensioned, z upward (check the L is above the XY plane). Contour Flange: pick the L, Distance 40. Leave the side the thickness goes on as it comes; note which way it went.	b: distance 50 Reference drawing (PNG)

	ID	MAKE	TWINS
☐	**S09** r2	Inventor only S01a + a Contour Flange on the edge at y = 0 (the Contour Flange command, not Flange): from that edge, 20 straight up, then 10 toward +Y, back over the plate. The result is a 20 tall wall along the whole 100 long edge with a 10 lip at its top folded back over the plate, so seen from the end (along X) the sheet is a C. Fig. 7 Reference drawing (PNG) How: Work Plane through the face's end at x = 0 (or sketch on that end face). Two lines starting at the corner of the edge at y = 0: up 20, then 10 toward +Y, dimensioned. Contour Flange, pick the profile, then the edge at y = 0, Edge extents. If Inventor can't create it, or the body disappears, try once with the material on the other side; if it still fails, save the file as it is, with the failed Contour Flange in the tree, and write "contour flange fails" on the card. That file still counts: the failed feature keeps its profile, edge, and settings.	b: the second line 15 instead of 10, so the lip is 15 Reference drawing (PNG)
☐	**S10**	Inventor only S01a + a second Face, then a Bend joining them. The second face: on a work plane parallel to XZ at y = 65, rectangle x 0…100, z 10…30, thickness toward −Y. How: Work Plane, Offset from Plane: XZ, 65. Sketch the rectangle, Face, flip the thickness toward the first face if needed. Bend command: pick the top edge of the first face at y = 60, then the bottom edge of the second face. Leave the bend options as they come.	none

	ID	MAKE	TWINS
☐	**S11**	High priority Inventor only S02a + Unfold the flange's bend, and save it unfolded. Reference drawing (PNG) How: Unfold command: Stationary Reference is the top face of the base (z = 2), then pick the bend. OK. Save while it is flat.	b: after the Unfold, Refold the same bend, then save Reference drawing (PNG)
☐	**S12**	Inventor only S11a + a cut across the bend while it is flat, then Refold. Open S11a, which is still flat: the base and the unfolded flange lie in one sheet that runs from y = 0 to about y = 80, with the bend zone just past y = 60. On that flat sheet, a rectangle x 40…60, y 50…66, so it straddles the bend line. Cut it through, then Refold. The result is the folded part with a 20 wide notch (x 40…60) through the bent corner: it takes 10 off the base's edge and the bottom of the flange. Reference drawing (PNG) How: Open S11a and Save As S12a while it is flat. Sketch on the flat top face (z = 2), rectangle dimensioned from the origin, x 40…60, y 50…66. Cut, Through All (in sheet metal, the Cut command). Then Refold the bend. Save it folded.	none

	ID	MAKE	TWINS
☐	**S13**	High priority S01a + a Hole Ø6 through at (10, 10). Reference drawing (PNG) How: Sketch on the top face, Point at (10, 10), dimensioned from the origin. Finish Sketch. Hole: Simple Hole, Drilled, Through All, diameter 6, on that point.	b: add a Rectangular Pattern of the hole: 3 along +X, spacing 15, so holes at x 10, 25, and 40 Reference drawing (PNG) · c: from **b**: add a Work Plane offset 50 from the YZ plane, then Mirror the hole and the pattern about it, so there are six holes (x 60, 75, and 90 too) Reference drawing (PNG) · d: from **c**: Create Flat Pattern, then Go To Folded Part, save Reference drawing (PNG)
☐	**S14**	High priority Inventor only S02a + Mirror the Flange across a work plane at y = 30, so a second flange stands on the edge at y = 0. How: Work Plane, Offset from Plane: the XZ Plane under Origin, offset 30 (check it sits at y = +30). Mirror (Sheet Metal tab): Features, the Flange, Mirror Plane the work plane.	none
☐	**S15**	High priority Inventor only S01a + Corner Round R3 on two corners of the face, (100, 60) and (0, 60), in one Corner Round. How: Corner Round command (sheet metal), click both corners, R3.	b: from a, a later Cut that removes the corner at (0, 60): sketch on the top face (z = 2) a rectangle x −5…10, y 50…65, dimensioned from the origin, Cut, Through All. The round at (0, 60) is gone; the one at (100, 60) stays

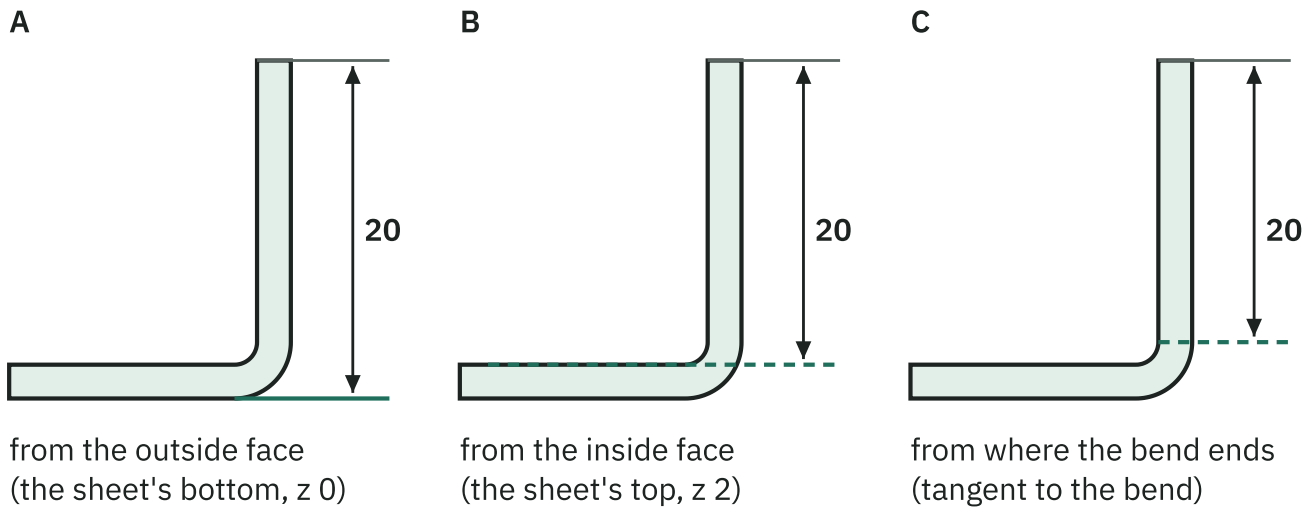


Fig. 3 · S02d / F08: the height datum. Side view along X; the flange stands at y = 60, thickness and bend drawn larger than scale. The height stays 20 in every case; only the face it is measured from changes. S02a: leave the default. S02d: pick a different one and write which (A, B, or C, or the app's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

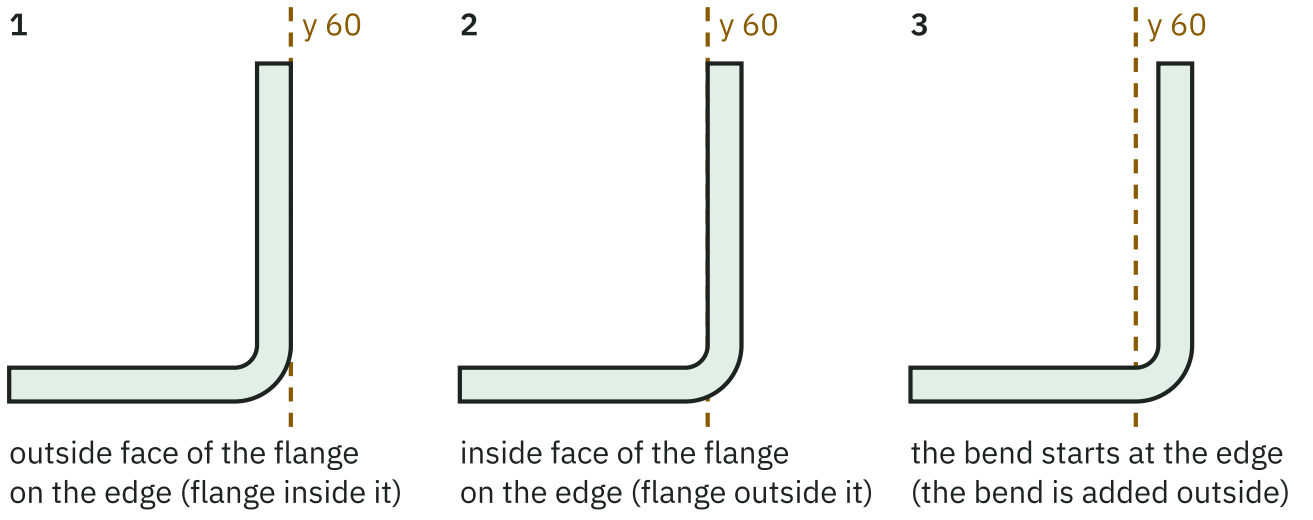


Fig. 4 · S02e: the bend position. The dashed line is the edge of the S01 face (y = 60) before the flange. The options move the flange relative to that edge. S02e: change only this, keep height 20, 90°, and the default height datum, and note which option.

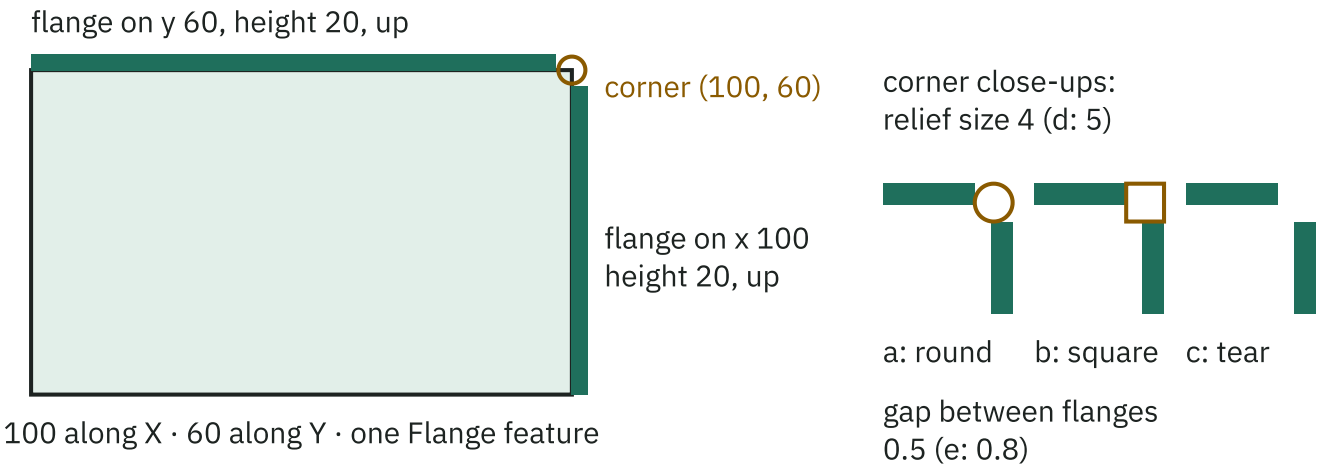


Fig. 5 · S03 corner, seen from above. One Flange feature with both edges picked, bend radius 1. The twins change only the corner: the relief shape, its size, or the gap. Each change goes in that flange's corner settings; check it shows before saving.

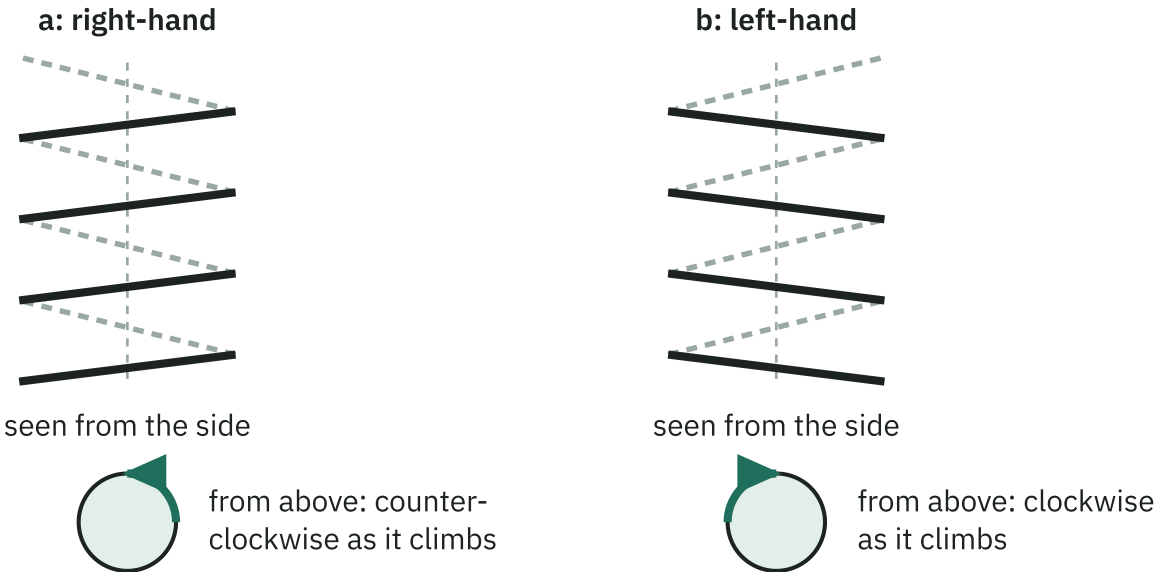


Fig. 6 · F13 coil hands. Solid lines are the front of the wire, dashed the back. a, right-hand (like a standard screw): looking down from above (+Z), the wire turns counter-clockwise as it climbs. b, left-hand: looking down from above, it turns clockwise as it climbs. In Fusion, a is the rotation checkbox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

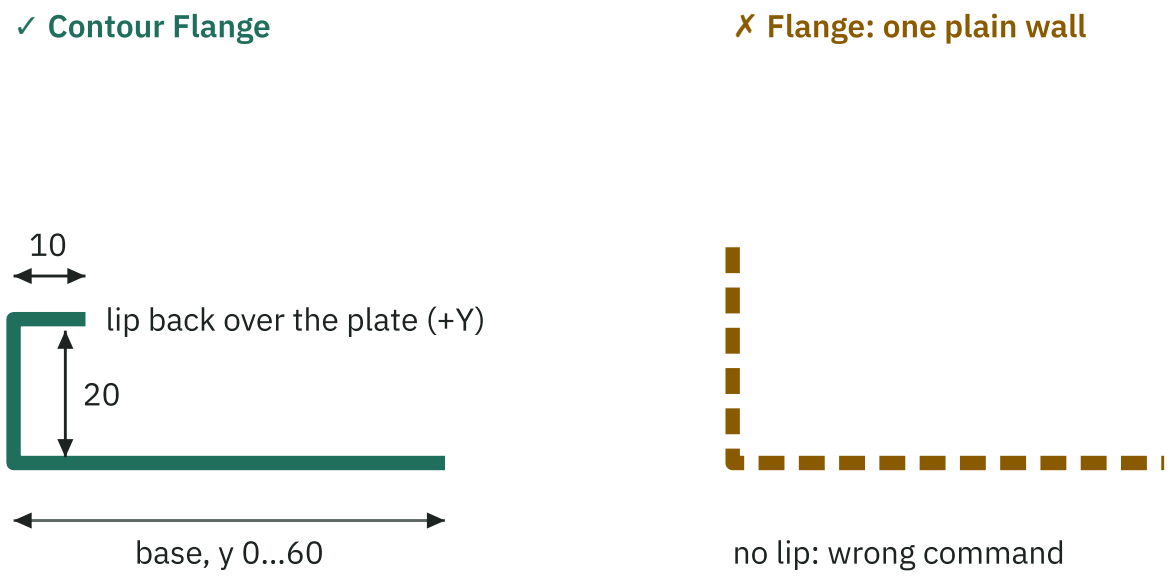


Fig. 7 · S09 seen from the end (along X). One Contour Flange on the edge at y = 0 makes the wall and the lip together, a C along the whole 100 edge. Flange gives a single plain wall: use Contour Flange.

Assemblies

Place P01a first, then P02a (and P03a / P16 where named). Keep the first part fixed in place (Inventor grounds it and SolidWorks fixes it by default) unless the twin says otherwise.

	ID	MAKE	TWINS
☐	**A01** r2	Place P01a and P02a, no constraints. Reference drawing (PNG) 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. Reference drawing (PNG) How: Constrain command, Mate type, face to face.	b: offset 5
☐	**A03**	A01a + one Flush: P02a bottom face to P01a bottom face, offset 0. Reference drawing (PNG) How: Constrain command, Mate type with Flush solution.	b: offset 5

	ID	MAKE	TWINS
☐	**A04**	A01a + one Angle constraint between P01a top face and P02a top face, 30°. Reference drawing (PNG) How: Constrain command, Angle type, Directed Angle, two faces.	b: 45°
☐	**A05**	P03a + P16. Insert: pin bottom circular edge into the hole top edge. Reference drawing (PNG) How: Constrain command, Insert type, hole edge to pin end edge.	b: offset 2
☐	**A06** r2	P01a + P16. Tangent: the pin's cylindrical face to P01a top face (z = 10). Reference drawing (PNG) How: Constrain command, Tangent type, cylinder face to planar face.	b: outside ↔ inside
☐	**A07** r2	A01a + rectangular component pattern of P02a, 3 × 1, spacing 70. How: Pattern Component, Rectangular tab.	b: 4 × 1
☐	**A08** r3	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
☐	**A09** r2	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. Reference drawing (PNG) 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** r2	Sketch on XY: rectangle 20 (X) × 30 (Y) centred on the origin, dimensioned. Extrude 10 upward (+Z). Reference drawing (PNG) 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** r2	User parameter `plate_w = 20 mm`; rectangle plate_w × 30, extrude `plate_w/2`. Reference drawing (PNG) How: Modify → Change Parameters, add a User Parameter. Type the names into the dimension and extrude distance.	b: plate_w = 25
☐	**F03** r3	F01a + fillet the vertical edge at x = +10, y = +15, radius 2 mm (R2; the corner edge, not a top edge; Fig. 1). Reference drawing (PNG) How: Fillet tool, select one edge, Constant radius.	b: R3 · c: edit a's fillet to a variable radius on the same vertical edge (x = +10, y = +15): R1 at its bottom end (z = 0, on the XY plane) and R3 at its top end (z = 10). Add no radius points between them and change nothing else. If Fusion asks how the radius changes from one end to the other, leave it as it comes and write what it shows on the card · d: from **c**: edit the same variable-radius fillet on the vertical edge at x = +10, y = +15, changing only the two radii: R3 at its bottom end (z = 0, on the XY plane) and R1 at its top end (z = 10). Add no radius points between them and change nothing else. Save As F03d

	ID	MAKE	TWINS
☐	**F04** r2	**High priority** F01a + hole Ø5 through, at the top face centre (0, 0, 10). Reference drawing (PNG) How: Hole tool (not an extrude cut), placed with a point on the top face at its centre, Simple, Through All.	b: counterbore Ø9 depth 3 · c: Extents Distance, depth 6, Drill Point Flat (a blind hole) · d: Extents To, picking the bottom face (z = 0) · e: from **c**: Drill Point Angle, 118° (Fusion's default tip), depth still 6
☐	**F05**	Plate x 0…60, y 0…40, z 0…5 + Ø4 hole at (5, 5), rectangular pattern 3 along +X × 2 along +Y, spacing 10. Reference drawing (PNG) How: Hole tool for one hole, then Rectangular Pattern, type Features, on that hole.	b: 4 × 2
☐	**F06** r3	Revolve: rectangle x 20–30, z 0–40 (above the XY plane), about Z, 360°. Fig. 2 Reference drawing (PNG) 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…100, y 0…60. How: Sheet Metal tab. Sheet Metal Rules, new rule with these values. Sketch the rectangle, Flange from the profile.	b: thickness 2.5 · c: from **a**: edit this design's rule (under Sheet Metal Rules, the one marked in use, not a library rule) and change only the K-factor to 0.3, then Save As F07c

	ID	MAKE	TWINS
☐	**F08** r3	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) · d: from **a**: edit the flange and change only its angle to 60°. If the Flange dialog has no angle, upload nothing for d and write that on the card · e: from **a**: edit this design's rule and change only the bend relief shape to Square. If the list has no Square, pick the shape nearest to it and write its name on the card · f: from **a**: as e, but the bend relief shape Tear (or the nearest, named on the card) · g: from **c**: edit this design's rule and change only the corner relief shape for a corner where two bends meet to Square (or the nearest, named on the card) · h: from **c**: as g, but Tear
☐	**F09** r2	F08c + Create Flat Pattern, finish, save. How: Create Flat Pattern, Finish Flat Pattern, save.	none

	ID	MAKE	TWINS
☐	**F10** r4	**High priority** Two components in one design (not separate part files): 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. Reference drawing (PNG) How: Appearance dialog, drag the paint onto the body; for b, onto the top face.	b: top face blue
☐	**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** r3	**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 Reference drawing (PNG) 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. Reference drawing (PNG) 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; if the Box won't snap to the point, make it with its first corner at the origin, then Modify → Move/Copy, Point to Point, from that corner to the sketch 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 (if it won't snap, make it at the origin and move it Point to Point onto the point), diameter 8, height 12 upward, so it spans x 1…9, y −11…−3, z 3…15

Feature history

Fusion only. One feature per file, made in the design history (keep the timeline on). Every sketch fully dimensioned from the origin. Leave every option the recipe doesn't name as Fusion shows it, and keep Fusion's names (Sketch1, Revolve1). If you had to change an option, write it on the card.

	ID	MAKE	TWINS
☐	F15	High priority Fusion only Revolve, one side, 90°: a rectangle x 20–30, z 0–40 on XZ, above the XY plane, about the Z axis. Fig. 2 Reference drawing (PNG) How: Sketch on XZ, rectangle x 20...30, z 0...40, dimensioned from the origin (Fig. 2 shows how to keep it above XY). Finish Sketch. Create → Revolve, New Body: Profile the rectangle, Axis the Z axis, Type Angle, Direction One Side, Angle 90. Either way round is fine.	b: Direction Two Sides, Angle 90 on side 1 and 45 on side 2 · c: Direction Symmetric, Angle 180 (the whole sweep, so 90 each way) · d: about a sketch line instead: in the same sketch, a line from (40, 0) to (40, 40), and the revolve's Axis that line, One Side, Type Full (360°) · e: new design: a rectangle x 0–10, z 0–40 on XZ (touching the Z axis) revolved about the Z axis, Type Full, so it makes a solid cylinder · f: Type To, and pick the YZ origin plane as the object, One Side

	ID	MAKE	TWINS
☐	**F16**	**High priority** **Fusion only** Copy of F01a, saved with no change. Each twin adds one construction plane to it; b and c also put a small block on theirs, so the volume shows where the plane went. Reference drawing (PNG) How: Open F01a and Save As F16a straight away. For each twin, open F16a, use Construct → the plane the twin names, with only the values it gives.	b: Offset Plane from the XY origin plane, distance 20; sketch on it a rectangle 20 × 30 centred on the origin, Extrude 5 up, New Body (it spans z 20…25) · c: Offset Plane from the block's top face, distance 5; sketch on it a rectangle 20 × 30 centred on the origin, Extrude 5 up, New Body (it spans z 15…20) · d: Plane at Angle on the top edge at y = +15 (the edge along X), angle 30 · e: Midplane between the top and bottom faces · f: new design: a cylinder Ø20 × 10 (circle Ø20 at the origin on XY, Extrude 10 up), then Tangent Plane on its side face, with the XZ plane as the reference and angle 0 · g: Plane Through Three Points: the top corners (−10, −15, 10) and (10, −15, 10), and the bottom corner (10, 15, 0) · h: Plane Along Path: the vertical edge at x = +10, y = +15, distance 0.5 (halfway up)

	ID	MAKE	TWINS
☐	F17	High priority Fusion only A cylinder Ø10 × 20 (circle Ø10 at the origin on XY, Extrude 20 up) with a cosmetic thread on its side face: ISO Metric profile, size 10, M10x1.5, class 6g, right hand, Full Length, Modeled off. How: Create → Thread, pick the side face. Thread Type ISO Metric profile, Size 10 mm, Designation M10x1.5, Class 6g, Direction Right hand. Leave Modeled unticked and Full Length ticked. Once the thread is on, Fusion makes the cylinder about Ø9.85, the thread's own diameter, even with Modeled unticked; that's expected.	b: Modeled ticked · c: Full Length unticked, Length 10, Offset 0 · d: Direction Left hand · e: new design: the F01a block (rectangle 20 × 30 centred on XY, Extrude 10 up), a Hole Ø8.5 through at the top-face centre (0, 0, 10), Simple, and the same thread on the hole's wall (cosmetic, Full Length) · f: Full Length unticked, Length 10, Offset 5 · g: Designation M10x1.25 (fine pitch), the rest as a
☐	F18	High priority Fusion only Copy of F01a, saved with no change. Each twin adds Joint Origins to it and nothing else. Reference drawing (PNG) How: Open F01a and Save As F18a straight away. For each twin, open F18a, use Assemble → Joint Origin, Type Simple unless the twin says otherwise, snap where it says, OK.	b: one Joint Origin snapped to the top-face centre (0, 0, 10) · c: a sketch on the top face with one Point at (5, 5), dimensioned from the origin, Finish Sketch, and a Joint Origin snapped to that point · d: one Joint Origin at the top-face centre with Angle 45 and Offset Z 5 · e: two Joint Origins: the top-face centre and the bottom-face centre (0, 0, 0) · f: Type Between Two Faces: the top and bottom faces, snapped to the centre, so it sits at (0, 0, 5)
☐	F19	High priority Fusion only A Base Feature holding a box x 0…20, y 0…30, z 0…10, made inside it. How: Solid → Create → Create Base Feature. Inside it, Create → Box: first corner at the origin, Length 20 along +X, Width 30 along +Y, Height 10 up. Finish Base Feature.	b: right-click the base feature in the timeline, Edit Feature, Press Pull the top face up 5 (it spans z 0…15), Finish Base Feature

	ID	MAKE	TWINS
☐	**F20**	High priority Fusion only A rectangle 20 × 30 centred on XY, Extrude with Direction Two Sides: 10 up and 4 down, so it spans z −4…10. Reference drawing (PNG) How: Sketch on XY, rectangle 20 × 30 centred on the origin, dimensioned. Finish Sketch. Create → Extrude, New Body, Direction Two Sides: Side 1 distance 10 (up), Side 2 distance 4 (down).	b: first add an Offset Plane 12 above XY; then edit the extrude: Direction One Side, Extent To Object, and pick that plane (it spans z 0…12) · c: Direction One Side, distance 10 up, Taper Angle −5 (the sides lean in, so the top is smaller) · d: Direction One Side, distance 10 up, Thin Extrude on, Thickness 2, Wall Location Center
☐	**F21**	High priority Fusion only F01a + Emboss: a rectangle 10 × 10 centred on the top face, raised 2 (it spans z 0…12). Reference drawing (PNG) How: Sketch on the top face, rectangle 10 × 10 centred on the origin, dimensioned. Finish Sketch. Create → Emboss: Profile the rectangle, Face the top face, Depth 2, Effect Emboss.	b: Effect Deboss (cut 2 deep) · c: instead of the rectangle, Text "AB" on the top face, centred, height 5, Emboss, Depth 1
☐	**F22**	High priority Fusion only F01a + a second body: a sketch on XY with a rectangle x 30…40, y 0…10, Extrude 10 up, New Body. Two separate bodies, nothing moved. How: Open F01a and Save As F22a. Sketch on XY, rectangle from (30, 0) to (40, 10), dimensioned from the origin. Finish Sketch. Extrude 10, Operation New Body.	b: Modify → Align, Objects Bodies: From the second body's bottom-face centre (35, 5, 0), To the first body's top-face centre (0, 0, 10). It then sits on top, x −5…5, y −5…5, z 10…20 · c: Modify → Move/Copy, the second body, Translate, Y Distance 20 (it spans y 20…30)

	ID	MAKE	TWINS
☐	**F23**	Fusion only Plate x 0…60, y 0…40, z 0…5 with a Ø4 hole through at (5, 5) cut by an extrude, then a Rectangular Pattern of the hole's faces: 3 along +X × 2 along +Y, spacing 10. How: Sketch on XY, rectangle (0, 0) to (60, 40), Extrude 5. Sketch on the top face, circle Ø4 at (5, 5), Extrude Cut through. Create → Pattern → Rectangular Pattern, Object Type Faces, pick the hole's wall, Direction 1 the X axis, Quantity 3, Distance Type Spacing, Distance 10; Direction 2 the Y axis, Quantity 2, Spacing 10.	none
☐	**F24**	Fusion only F01a + a Rectangular Pattern of the body: 3 along +X, spacing 30, so three blocks. Reference drawing (PNG) How: Open F01a and Save As F24a. Create → Pattern → Rectangular Pattern, Object Type Bodies, pick the body, Direction the X axis, Quantity 3, Distance Type Spacing, Distance 30, one direction only.	none
☐	**F25**	Fusion only F01a + a chamfer with two distances on the top edge along X at y = +15, z = 10: 1 and 2, the sides as Fusion puts them. `Fig. 1` How: Open F01a and Save As F25a. Modify → Chamfer, pick that one edge, Chamfer Type Two Distances, Distance 1 = 1, Distance 2 = 2. Don't press Flip and change nothing else. Write on the card which face got the 2: the top face or the +Y end face.	b: from **a**: edit the chamfer, press Flip once, so the 1 and the 2 swap faces, OK. Change nothing else, and write on the card which face got the 2 · c: edit a's chamfer to Distance and Angle on the same edge: Distance 2, Angle 30°. Leave the sides as they come (don't press Flip) and change nothing else. Write on the card which face the 2 runs along: the top face or the +Y end face · d: from **c**: edit the chamfer, press Flip once, so the 2 runs along the other face, OK. Change nothing else, and write on the card which face the 2 runs along

	ID	MAKE	TWINS
☐	**F26**	Fusion only Three sketches in three places: one in the root, one in a component Bracket, one in a component Pin. Root: a sketch on XY with a circle Ø8 at (−15, 10), not extruded. Bracket: a sketch on XY with a rectangle x 0…40, y 0…20, Extrude 5 up (it spans z 0…5). Pin: a sketch on XY with a circle Ø6 at (60, 10), Extrude 15 up (it spans x 57…63, y 7…13, z 0…15). Nothing moved, no joints. How: New design. With the root active, Create Sketch on XY, circle Ø8 centred at (−15, 10), dimensioned from the origin, Finish Sketch. Assemble → New Component, Type Empty, Name Bracket, Activate ticked, OK. With Bracket active (its dot filled in the browser), expand Bracket → Origin in the browser and sketch on its XY: rectangle from (0, 0) to (40, 20), dimensioned. Finish Sketch. Extrude 5, New Body. Activate the root (the dot on the top row of the browser), then New Component again, Name Pin, Activate ticked, and in the same way sketch on Pin's own XY: circle Ø6 centred at (60, 10), dimensioned from the origin. Finish Sketch. Extrude 15, New Body. Activate the root again before saving. Check the browser: the root, Bracket, and Pin each show one sketch in their own Sketches folder; leave the names Fusion gives them.	none
☐	**F27**	Fusion only F01a + fillet the vertical edge at x = +10, y = +15, R2, as in F03a, but with curvature (G2) continuity and a Tangency Weight of 0.5. Fig. 1 How: Open F01a and Save As F27a. Modify → Fillet, pick that one edge, Radius 2. Set Continuity to Curvature (G2), then set Tangency Weight to 0.5. Change nothing else. If your Fusion names these differently, use the closest and write what it shows on the card.	b: Tangency Weight 1.5 · c: Tangency Weight left at its default with Curvature (G2) set; write the default on the card

	ID	MAKE	TWINS
☐	F28	**High priority** Two components, no joints. Component1 holds one body: a rectangle 20 × 30 centred on the origin, extruded 10 (as F01a). Component2 holds two bodies: a cylinder Ø16 centred at (40, 0), extruded 15, and a block from (60, −5) to (70, 5), extruded 5. Everything shown. How: New Component, then the sketch and Extrude inside it. Activate the top-level design, New Component again, and make both of its bodies inside it: the block as a New Body, not a Join. Keep each eye icon on. Each twin changes only one eye icon in the browser.	b: hide Component2 (its eye icon in the browser) · c: keep Component2 shown, and hide only its cylinder body (the eye icon on that body under Component2 → Bodies)
☐	F29	**Fusion only** Two overlapping circles in one sketch, and one region of them extruded 10 as a New Body. Circle A: Ø40 centred on the origin. Circle B: Ø12 centred at (0, −20), so it crosses A and makes three regions: A-only (inside A, outside B), the lens (inside both), and B-only (inside B, outside A). a extrudes A-only: a Ø40 disk with a notch. How: New design. Create Sketch on XY. Circle A: centre at the origin, Ø40. Circle B: centre at (0, −20), Ø12. Dimension the two circles and B's centre, nothing else. Finish Sketch. Extrude, Distance 10, Operation New Body, and click inside only the A-only region (the big area, not the small lens where the circles overlap). Don't edit the sketch after the extrude. Start each twin from a new, empty design, the same way.	b: select only the lens · c: select only B-only (the small crescent outside A) · d: select A-only and the lens together (all of circle A)

	ID	MAKE	TWINS
☐	**F30**	Fusion only An inserted part below two components. L-part is a separate design, an L-shaped block spanning x 0…40, y 0…20, z 0…10. In a, the new design has two empty components at the root, Owner A and Owner B, with one L-part inserted in each. Owner A stays at the origin. Owner B turns 180° about the root Z axis, then moves to (110, 130, 170). Each L-part sits at (10, 0, 0) inside its owner, so the corner that started at L-part's origin ends at (10, 0, 0) in Owner A and at (100, 130, 170) in Owner B. How: L-part first. New design, Create Sketch on XY, lines (0, 0) → (40, 0) → (40, 10) → (10, 10) → (10, 20) → (0, 20) and back to the start, fully dimensioned from the origin. Finish Sketch, Extrude 10 up, New Body. Save it to the cloud as L-part. Then F30a: new design, saved to the cloud. Assemble → New Component, Type Empty, named Owner A. Activate the root again (the dot beside the top item in the browser), then New Component again, named Owner B, so both sit directly under the root, side by side, not one inside the other. Activate Owner A, right-click L-part in the Data Panel, Insert into Current Design, enter X 10, and OK. Do the same with Owner B active. Activate the root. If a move later says the components can't move, right-click Owner A and Owner B and Unground them. Select Owner B in the browser, Modify → Move/Copy, Move Object Component, Set Pivot at the root origin, Z Angle 180, OK. Move/Copy again, X 110, Y 130, Z 170, OK. Capture Position: click it where your Fusion shows it after a move (the Position group, or the timeline); if it has none, leave the position as it is and write that on the card. Check both corners with Inspect → Measure. Look at the whole design once with everything shown, then save. Export with File → Export → .f3z, so the inserted parts travel inside it, and send only the .f3z. If your Fusion names a step differently or won't do it, write what you did on the card.	b: move Owner A by (20, 30, 50) with Move/Copy, then Capture Position as in a, so its L-part's corner is at (30, 30, 50)

	ID	MAKE	TWINS
☐	F32	High priority Fusion only A box with two flanges: the box x −20…20, y −20…20, z 0…30, and two 3 mm flanges at its foot, x 20…30 and x −30…−20, each y −20…20, z 0…3, extruded from the box's bottom face. How: New design. Create Sketch on XY, Center Rectangle 40 × 40 on the origin. Finish Sketch, Extrude 30 up, New Body. Create Sketch on the top face (z = 30): two rectangles, x 20…30 and x −30…−20, each y −20…20, dimensioned from the origin. Finish Sketch. Extrude both rectangles: Start From Object, click the box's bottom face (z = 0); Extent Distance 3, going up (+Z); Operation Join. Keep every feature in the timeline.	b: a + Fillet R4 on the four vertical box edges at x = ±20, y = ±20 (above the flanges, z 3…30) · c: b + Chamfer, Equal Distance 0.5, on the 8 edges round the top face (z = 30): 4 straight and 4 rounded. Click the 8 edges, not the face
☐	F33	High priority Fusion only A plate with two walls, from one sketch. Sketch on XY: a square 40 × 40 centred on the origin, a circle Ø15 at the origin, lines x = −17 and x = 17 from y = −20 to y = 20 (making a 3 mm strip along each side), and short lines at y = 0 across each strip, from (−20, 0) to (−17, 0) and from (17, 0) to (20, 0). Extrude1: the plate between the strips, outside the circle, 3 up. Extrude2: the four strip pieces, 15 up, as walls. How: New design. Create Sketch on XY and draw all of it in one sketch, dimensioned from the origin. Finish Sketch. Extrude1: click the one region between the strips and outside the circle (the circle stays a hole), Distance 3, New Body. Show Sketch1 again (the eye in the browser), Extrude2: click the four strip pieces, Distance 15, Join. Keep every feature in the timeline.	b: the same without the two short lines at y = 0, so each strip is one piece; Extrude2 takes the two strips

	ID	MAKE	TWINS
☐	F34	**High priority** **Fusion only** An X-braced window cut through a block. Block x −30…30, y −20…20, z −5…5. On XY, a window 40 × 24 centred on the origin with R4 corners, and two crossing ribs that join its top and bottom edges. Rib 1: lines (−11.5, −12) → (8.5, 12) and (−8.5, −12) → (11.5, 12). Rib 2: lines (11.5, −12) → (−8.5, 12) and (8.5, −12) → (−11.5, 12). One cut through all, symmetric, takes the four openings around the ribs: left, right, top, and bottom. How: New design. Create Sketch on XY, Center Rectangle 60 × 40. Finish Sketch, Extrude Symmetric, 5 on each side (z −5…5), New Body. Create Sketch on XY again: Center Rectangle 40 × 24, Sketch Fillet R4 on its four corners, then the four rib lines with their ends on the window's top and bottom edges, dimensioned from the origin. Finish Sketch. Extrude: click the four openings (not the ribs or the small diamond where they cross), Extent All, Direction Symmetric, Operation Cut. Keep every feature in the timeline.	none
☐	F35	**High priority** **Fusion only** Four countersunk holes with a drill point. Plate x −20…20, y −20…20, z 0…10. On the top face, four sketch points at (±10, ±10). One Hole feature at all four points: Countersink, Ø5, countersink Ø10 at 90°, depth 6, Drill Point Angle 118°. How: New design. Create Sketch on XY, Center Rectangle 40 × 40, Extrude 10 up, New Body. Create Sketch on the top face: Point four times, dimensioned from the origin. Finish Sketch. Create → Hole, Placement From Sketch (Multiple), click the four points. Hole Type Countersink, Diameter 5, Countersink Diameter 10, Countersink Angle 90°, Extents Distance, Depth 6, Drill Point Angle 118°. Keep every feature in the timeline.	b: Drill Point Flat, everything else as a · c: one point only, at (10, 10), everything else as a

	ID	MAKE	TWINS
☐	F36	High priority Fusion only F01a + one Fillet feature with two radius groups that meet at the corner (10, 15, 10): R2 on the top edge along X at y = +15, z = 10, and R3 on the vertical edge at x = +10, y = +15. Fig. 1 How: Open F01a and Save As F36a. Modify → Fillet, click the top edge at y = +15, Radius 2. Click + to add a second selection set, click the vertical edge at x = +10, y = +15, Radius 3. OK, one feature.	b: the same finished shape on purpose, made as two features: from **a**, Save As F36b, delete a's Fillet feature, then make a new Fillet on the top edge at y = +15, R2, and then another new Fillet on the vertical edge at x = +10, y = +15, R3. The difference is in the timeline, not the part

	ID	MAKE	TWINS
☐	F37	High priority Fusion only Two components, A and B, nothing moved, no joints. A: a box x 0…40, y 0…20, z 0…10. B: a plate x 10…30, y 0…20, z −5…0, and a cylinder Ø8 centred at (20, 10) whose Extrude starts From Object on A's top face (z = 10) and goes 15 up, so it spans z 10…25. How: New design. Assemble → New Component, Type Empty, Name A, Activate ticked, OK. With A active, expand A → Origin in the browser and sketch on its XY: rectangle from (0, 0) to (40, 20), dimensioned. Finish Sketch. Extrude 10 up, New Body. Activate the root, then New Component again, Name B, Activate ticked. Sketch on B's own XY: rectangle from (10, 0) to (30, 20), dimensioned. Finish Sketch. Extrude 5 down (z −5…0), New Body. With B still active, a new sketch on B's XY: circle Ø8 centred at (20, 10), dimensioned from the origin. Finish Sketch. Extrude the circle: Start From Object, click A's top face (z = 10); Extent Distance 15, going up (+Z); Operation New Body. Check the browser: A holds one sketch and one body, and B holds two sketches and two bodies, the cylinder among them. Activate the root before saving.	b: from a, edit the cylinder's Extrude and change only its Start face: A's bottom face (z = 0) instead of its top face, still 15 up, so the cylinder spans z 0…15 · c: from a, the same edit with B's own plate as the Start face: the plate's bottom face (z = −5), so the cylinder spans z −5…10 · d: from a, after the cylinder, activate A and add Fillet R2 on the four edges round A's top face (z = 10), so a later feature of A changes the face the cylinder starts on. The cylinder's Extrude stays as it is; activate the root before saving · e: from a, a second copy of A: activate the root, right-click A in the browser, Copy, then Paste, type Z 20 in the move box, OK, so the copy spans z 20…30. Leave the names Fusion gives (A:1 and A:2). The cylinder still starts on the first A's top face · f: from **e**: edit the cylinder's Extrude and change only its Start face, to the top face of A:2 (z = 30), so it spans z 30…45 · g: from a, a Combine in B with a tool body from A: activate B, Modify → Combine, Target Body B's plate, Tool Bodies A's box, Operation Join, Keep Tools left as it comes (note what it showed). The plate and the

	ID	MAKE	TWINS
			box touch at z = 0, so they join into one body. Activate the root before saving
☐	F38	High priority Fusion only F07a + two separate Flange features: one on the top edge at y = 60, height 20, angle 90°, and one on the edge at y = 0, height 20, angle 45°, both going up. How: Open F07a and Save As F38a. Flange on the edge at y = 60: Height 20, angle 90, OK. Then a second Flange on the edge at y = 0: Height 20, angle 45, OK. If the Flange dialog has no angle, upload nothing and tell us in the chat.	b: from **a**: edit the second flange (y = 0) and change only its angle to 120°
☐	F39	High priority Fusion only A flange made from an open sketch: an L on XZ from (0, 0) to (50, 0) to (50, 30), above the XY plane, made 40 wide along +Y. The sheet metal rule as in F07a (thickness 2, bend radius 1, gap 0.5). How: New design, units mm. Sheet Metal tab, a rule as in F07a. Create Sketch on XZ: two Lines, (0, 0) to (50, 0) and (50, 0) to (50, 30), dimensioned from the origin, left open. Finish Sketch. Flange: pick the open L, Distance 40 going +Y, OK. Fusion bends it at the corner. If the Flange tool won't take an open sketch, upload nothing and tell us in the chat.	b: from **a**: edit the sketch and change only the 30 to 20
☐	F40	High priority Fusion only F08a + a hem on the flange's free (top) edge, folded toward −Y. How: Only if your Fusion has a Hem command on the Sheet Metal tab. Open F08a and Save As F40a. Hem on the flange's top edge, folded toward −Y, its other values as they come. If there is no Hem command, skip F40 and tell us in the chat.	none

	ID	MAKE	TWINS
☐	**F41**	High priority Fusion only F09a with its flat pattern exported as a DXF. How: Open F09a and Save As F41a. Open the Flat Pattern (double-click it in the browser), then right-click it and Export DXF, keeping the default options. Upload the .f3d with the .dxf beside it.	none

SolidWorks

Make the Inventor part, sheet-metal, assembly, and drawing recipes above and below (P, S, A, and D IDs, except those marked Inventor only) with the SolidWorks tools below, and save them as `P01a.sldprt`, `S02b.sldprt`, `A02a.sldasm`, `D05a.slddrw`. 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. SolidWorks shows no Z axis (P02): Reference Geometry → Axis, pick the Front and Right planes, and revolve about it. +Z can look downward in the isometric view; the ring must end at z 0…40. For P02b, write which quarter is missing
Extrude · Revolve · Sweep · Loft	Extruded Boss/Base · Revolved Boss/Base · Swept Boss/Base · Lofted Boss/Base
Hole command (P03, P07, P08, D04)	Hole Wizard, Hole type "Hole" (the plain one, not Dowel), with the diameter and end condition the recipe gives (P03a: Ø5, Through All), placed on a point on the face, fully dimensioned. Only P03b uses Counterbore (Ø9, depth 3)

RECIPE SAYS	IN SOLIDWORKS
Fillet · Chamfer · Shell	Fillet (Constant size) · Chamfer (Angle-Distance, 45°, or Distance-Distance with Equal distance ticked) · Shell. P04d, e: SolidWorks cannot change a fillet's type while editing it, so delete Fillet1 and add a new Fillet, Fillet Type: Variable Size, on the same edge (R1 at the z 0 end, R3 at the z 10 end for d, swapped for e, no extra points); write on the card that you remade it, and which transition (Smooth or Straight) it shows. P05d: SolidWorks has no Flip for Distance-Distance, so edit c's chamfer and swap the values (Distance 1 = 2, Distance 2 = 1), and write which face has the 2
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. Volume check for P14a: 299.43 (SolidWorks may show up to about 299.5; within 0.1 is fine)
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: Thickness 2, Bend Allowance K-factor 0.44. Then right-click the Sheet-Metal feature in the tree, Edit Feature: Default bend radius 1, Auto Relief on. SolidWorks has no round relief of size 4 and no corner gap 0.5 for a plain plate: leave them as they come and say so on the card
Face (S01) · Flange (S02, S03) · Hem (S04)	Base Flange/Tab · Edge Flange (S03: both edges in one Edge Flange) · Hem (Closed/Open, gap 1)

RECIPE SAYS	IN SOLIDWORKS
Height datum · bend position (S02d/e)	Edge Flange → Flange Length reference (Outer/Inner virtual sharp, Tangent) · Flange Position (Fig. 4: 1 = Material Inside, 2 = Material Outside, 3 = Bend Outside; Bend From Virtual Sharp and Tangent To Bend have no picture here, leave them alone). S02i has no SolidWorks equivalent (Edge Flange has no Bend Transition): skip it
Corner relief / gap (S03) · Corner Round (S07)	Corner Relief, its own feature after the Edge Flange (Sheet Metal → Corners → Corner Relief, 2 Bend Corner: a Circular, b Rectangular, c Tear, size 4, d 5); the gap where the walls meet is the Edge Flange's Gap distance (Flange Parameters, shown once two edges that meet are picked; or Corners → Closed Corner): 0.5, e 0.8; f (the Bend tab elsewhere): the Edge Flange's Custom Relief Type, Obround, sizes as they come, Corner Relief untouched · 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
Insert (A05) · Tangent (A06)	Concentric + Coincident on the edges (b: Distance 2) · Tangent mate (A06a: pin rests on top of the face, axis 2.5 above z 10; b: pin inside)
Pattern Component (A07)	Open A01a, File → Save As A07a (do not use Insert Component), then Linear Component Pattern of P02a, 3 × 1, spacing 70
Drawing (D05)	New → Drawing from the default template, sheet format as it comes. Model View of D04a showing the L, scale 1:1, then Projected View for the other two. Smart Dimension for the four values. Save as `D05a.slddrw` beside D04a.sldprt, then File → Save As → Dxf as `D05a.dxf` (Options: R2018 or R2000, scale 1:1, fonts as text)

RECIPE SAYS	IN SOLIDWORKS
Model state (A08) · Display State (A09)	A08: Save As A08a, then Configurations → Add Configuration, suppress P02a in the new one, activate Default, and save (b: save with the new one active) · A09: Save As A09a, then in the ConfigurationManager add a Display State named View1 with P02a hidden, keep the default display state with both parts visible, activate the default, and save (b: save with View1 active)

Creo

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs, except those marked Inventor only) with the Creo tools below, and save them as `p01a.prt`, `s02b.prt`, `a02a.asm`. Start from the millimetre templates (`mmns_part_solid_abs`, `mmns_part_sheetmetal_abs`, and `mmns_asm_design_abs`; untick Use default template to pick them). Creo keeps names in lower case and numbers each save (`p01a.prt.3`): that's fine, send the newest one. For a twin, open its a file, File → Save As → Save a Copy under the twin's name, then open the copy and make the one change. Each release is its own set, since a newer release's files don't open in an older one. 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 CREO
Planes XY · XZ · YZ (coordinates as written)	FRONT · TOP · RIGHT datum planes. Creo shows Y up; keep the recipe's x, y, z values as they are, so "upward (+Z)" means toward the viewer from FRONT
Extrude · Revolve · Sweep · Loft	Extrude · Revolve · Sweep (constant section) · Blend
Hole command (P03, P07, P08)	Hole tool: Simple, Through All, placed on the face with linear references to two datum planes (P03b: standard profile, Add counterbore Ø9, depth 3)
Fillet · Chamfer · Shell	Round (constant) · Edge Chamfer (D × D) · Shell
Rectangular / Circular Pattern (features) · Mirror	Pattern, Direction type (both directions) · Pattern, Axis type, about the Z axis · Mirror (feature, about the datum plane the recipe names, mapped as in the first row)

RECIPE SAYS	IN CREO
Work plane offset · normal to curve	Datum Plane: Offset from a plane · Sweep sketches its section at the start of the path, normal to it, so P10 needs no extra plane
Loft between two profiles (P11)	Blend, Selected Sections: the square and the circle. Divide the circle into 4 arcs so both sections have 4 segments, with matching start points
User parameter plate_w (P12)	Parameters: add a Real Number `plate_w = 20`, then Relations: the rectangle's width dimension = `plate_w`, the extrude depth = `plate_w / 2`
3D sketch (P14)	Datum Points (offset from the coordinate system) at the three points, a Datum Curve Through Points with straight segments, then Sweep a Ø3 circle along it
Appearance (P15, A10)	Appearance gallery: part red; b: top face blue · in the assembly, the component's appearance blue
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	Model Properties → Sheetmetal: thickness 2, inside bend radius 1, K factor 0.44, corner relief round (or the closest shape) size 4, corner gap 0.5 where offered; note what you set
Face (S01) · Flange (S02, S03) · Hem (S04)	Planar wall · Flange wall (S03: both edges as one chain in one Flange wall) · Flange wall with a hem shape, gap 1; note which shape
Height datum · bend position (S02d/e)	Flange wall: the height reference on its Shape tab · the wall's position against the edge (Offset tab); note which option
Corner relief / gap (S03) · Corner Round (S07)	Flange wall Relief tab, corner relief and gap · Round on the corner's edge through the thickness, R3
Flat Pattern, back to folded (S05)	Flat Pattern feature, then Suppress it so the part is folded, save
Ground / unground (A01)	Default constraint (fixed in place) · left Packaged with no constraints

RECIPE SAYS	IN CREO
Mate · Flush · Angle (A02–A04)	Coincident, faces opposed (b: Distance 5) · Coincident, faces aligned · Angle Offset
Insert (A05) · Tangent (A06)	Coincident axes plus Coincident faces on the hole's top (b: Distance 2) · Tangent
Pattern Component (A07)	Pattern of the component, Direction type, 3 members, spacing 70
Model state (A08) · View representation (A09)	Family Table: a second instance with P02a suppressed (save with the generic active) · View Manager: a Combined State "View1" with P02a hidden

CATIA

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs, except those marked Inventor only and the omissions below) with the CATIA tools in this table. In CATIA V5, save each part as `P01a.CATPart` or `S02b.CATPart`, and each assembly as `A02a.CATProduct`. In CATIA V6 / 3DEXPERIENCE, export each part or assembly as a CATIA V5 file: Export, format CATPart / CATProduct, and pick the V5 release it targets. Choose that target release in the portal, for example V5-6R2024, and record both versions, for example “3DEXPERIENCE R2024x, saved as V5-6R2024”. Each V5 release is its own set of files. Keep the referenced CATPart files beside each CATProduct. Use millimetres, fully constrain sketches, and make every twin from its a file with just the one change. Leave out P14 (3D sketch), A08 (model states), and A09 (view representations): these have no plain equivalent here. If another option differs, note what you used on the card. Check volume with Measure Inertia (Measure toolbar).

RECIPE SAYS	IN CATIA
Planes XY · XZ · YZ (coordinates as written)	xy · zx · yz planes; keep the recipe's x, y, z coordinates and positive directions. Part Design → Sketcher for the profiles, with dimensional and geometric constraints
Extrude (P01, P07–P09, P12, P13, P16)	Part Design → Pad; use a circle for the discs and pin, and a sketch on the end face for the boss

RECIPE SAYS	IN CATIA
Revolve (P02)	Part Design → Shaft, 360° about the sketch axis; keep the ring above XY
Hole command (P03, P07, P08)	Part Design → Hole, Up To Last, positioned with a constrained positioning sketch (P03b: Counterbored, Ø9, depth 3)
Fillet (P04) · Chamfer (P05) · Shell (P06)	Part Design → Edge Fillet, constant radius · Chamfer, Length1/Angle, 45° · Shell, remove the top face, inside thickness
Rectangular / Circular Pattern (P07, P08) · Mirror (P09)	Part Design → Rectangular Pattern, Instances & Spacing in both directions · Circular Pattern about Z, complete crown · Mirror of the boss about zx
Sweep (P10)	Part Design → Rib with the circle as profile and the rounded sketch path as center curve; Sketcher → Corner for the path's R10. Plane, Normal to curve, at the path start for the circle
Loft (P11)	Part Design → Multi-sections Solid between the square and circle sketches; use matching closing points and note the coupling option
User parameter plate_w (P12)	Formula: create a Length parameter `plate_w = 20mm`; link the sketch width to `plate_w`, and the Pad length to `plate_w / 2`
Work plane offset (P11, P13)	Plane, Offset from plane, from xy; sketch on the resulting plane
Appearance (P15, A10)	Properties → Graphic → Color on the part or face; in Assembly Design, set the component instance's color for A10
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	Generative Sheetmetal Design → Sheet Metal Parameters: thickness 2, bend radius 1, K factor 0.44; set round corner relief size 4 and gap 0.5 where offered, and note the settings
Face (S01) · Flange (S02, S03)	Generative Sheetmetal Design → Wall from the rectangle · Wall On Edge, height 20, angle 90°. For S03, select both edges in the same feature; note any limitation of the release
Height datum · bend position · width (S02)	Wall On Edge: change the length reference, wall position, or limits to match the named twin, keeping the other values. Record the option used

RECIPE SAYS	IN CATIA
Hem (S04)	Generative Sheetmetal Design → Hem on the flange's free edge, toward −Y, gap 1 (b: 1.5)
Corner relief / gap (S03) · Bend relief (S06)	Sheet Metal Parameters and the wall's bend and corner options: use the named relief shape, size, width, or gap; record the option used
Flat Pattern, back to folded (S05)	Generative Sheetmetal Design → Unfolding, then return to the folded view before saving
Corner Round (S07)	Generative Sheetmetal Design → Corner, on the named wall corner, radius 3; c: all four corners in the same feature
Place parts · Ground / unground (A01)	Assembly Design → Existing Component to insert P01a and P02a · Fix on each component for a; remove the Fix constraints for b
Mate · Flush · Angle (A02–A04)	Assembly Design → Contact with opposed orientation (A02b: Offset 5) · Coincidence with aligned orientation · Angle constraint; select the recipe's faces and record the orientation
Insert (A05) · Tangent (A06)	Assembly Design → Coincidence of the axes plus Contact of the seating faces (A05b: Offset 2) · Contact between the cylindrical and planar faces, with the named inside or outside orientation
Pattern Component (A07)	Assembly Design → Define Multi Instantiation of P02a, direction +X, 3 instances, spacing 70 (b: 4); keep P01a in place

FreeCAD

These recipes use what only FreeCAD documents hold: sketch curves, expressions and spreadsheets, Part containers, groups, and links, PartDesign transforms, ShapeBinders, and every Sketcher constraint. We make them ourselves in FreeCAD 0.21.2, 1.0.2, and 1.1.4 from the scripting API, so there is nothing to send. Work in millimetres; each recipe starts a new document with one Body unless it says otherwise.

	ID	MAKE	TWINS
☐	**N01**	On XY, an ellipse centred at (0, 0), with semi-major radius 20 along +X and semi-minor radius 10. Pad 5 up. Reference drawing (PNG) How: Block the ellipse to fix its centre, radii, and axis.	b: rotate only the major axis 30° counterclockwise
☐	**N02**	On XY, an ellipse with centre (0, 0), semi-major radius 20 along +X, and semi-minor radius 10. Keep the arc from parameter 0 to π/2, and close it with its chord. Pad 5 up. Reference drawing (PNG) How: Block the arc and chord. Parameters follow (20 cos t, 10 sin t).	b: use the complementary arc from π/2 to 2π with the same chord
☐	**N03**	On XY, a non-rational quadratic B-spline with poles (0, 0), (10, 20), and (20, 0), knots 0 and 1, and multiplicities 3 and 3. Close it with a line. Pad 5 up. Reference drawing (PNG) How: Block the spline and line.	b: elevate the spline degree to 3 without changing its shape
☐	**N04**	On XY, a periodic non-rational quadratic B-spline with cyclic poles (−10, −10), (10, −10), (10, 10), and (−10, 10). Knots 0, 1, 2, 3, and 4 each have multiplicity 1. Pad 5 up. Reference drawing (PNG) How: Block the closed spline.	b: pad length 6 · c: the same shape made clamped (non-periodic): poles (−10, −10), (10, −10), (10, 10), (−10, 10), and (−10, −10) again, knots 0, 1, 2, and 3 with multiplicities 3, 1, 1, and 3, closed by its own ends
☐	**N05**	On XY, the hyperbola arc (10 cosh t, 5 sinh t), from t = −ln 2 to ln 2, closed by its chord. Pad 5 up. Reference drawing (PNG) How: Block the arc and chord to fix the centre, axes, radii, and endpoints.	b: change only the symmetric parameter limit from ln 2 to ln 3
☐	**N06**	On XY, a parabola with vertex (0, 0), axis +X, and focal length 2. Trim at y = −8 and 8, and close with the chord at x = 8. Pad 5 up. Reference drawing (PNG) How: Use $x = y^2/8$. Block the arc and chord.	b: focal length 4, keeping the y limits and updating the chord

	ID	MAKE	TWINS
☐	**N07**	On XY, the right half of the ellipse (20 cos t, 10 sin t), from t = −π/2 to π/2. Close through (−20, 10) and (−20, −10) with three lines. Pad 5 up. Reference drawing (PNG) How: Block the arc and each line.	b: ellipse semi-major radius 25
☐	**N08**	On XY, a circle centred at (0, 0), radius 10. Pad 5 up. Reference drawing (PNG) How: Block the circle.	b: circle radius 12
☐	**N09**	On XY, a radius 10 circle centred at (0, 0). Keep the arc from 0 to π/2 counterclockwise from +X, and close with its chord. Pad 5 up. Reference drawing (PNG) How: Block the arc and chord.	b: use the complementary arc from π/2 to 2π with the same chord
☐	**N10**	A rectangle from (0, 0), width 40, height 20, padded by half its width constraint. Reference drawing (PNG) How: Name the width constraint width. Set Pad Length to Sketch.Constraints.width / 2.	b: width constraint 50
☐	**N11**	A rectangle from (0, 0), width 30, height 20, padded 10. A spreadsheet alias drives its width. Reference drawing (PNG) How: Set Dimensions.B1 to 30 mm with alias w. Set the sketch width expression to Dimensions.w.	b: spreadsheet B1 45 mm
☐	**N12**	A Body with a 10 × 20 rectangle padded 5 inside a Part container. Reference drawing (PNG) How: Start the rectangle at (0, 0). Place the Body at (2, 3, 0). Place the Part at (100, 20, 5), rotated 90° about Z. Measure in global coordinates.	b: Part position (120, 20, 5)

	ID	MAKE	TWINS
☐	**N13**	A document group holding two Bodies, each a 10 × 20 rectangle padded 5. Reference drawing (PNG) How: Start both rectangles at (0, 0). Place the Bodies at (2, 3, 0) and (32, 3, 0). Count both solids, including hidden Bodies.	b: hide the second Body
☐	**N14**	A Body and a link to it, each counted once: two solids, total volume 2000. Reference drawing (PNG) How: Pad a 10 × 20 rectangle from (0, 0) by 5. Place the Body at (2, 3, 0), and its link at (32, 3, 0). Include the source and the instance in the bounds.	b: link position (52, 3, 0)
☐	**N15**	Two Part containers, each holding a Body with a 10 × 20 rectangle padded 5. Reference drawing (PNG) How: Start both rectangles at (0, 0). Place the Bodies at (2, 3, 0) and (32, 3, 0), and their Parts at (100, 0, 0) and (200, 0, 0). Count both solids in global coordinates.	b: second Part position (250, 0, 0)
☐	**N16**	A 40 × 30 plate from (−20, −15), padded 4, with a mirrored boss. Reference drawing (PNG) How: On z = 4, pad a 5 × 5 rectangle from (5, 5) by 6. Mirror only the boss about YZ.	b: mirror plane XZ
☐	**N17**	A 40 × 60 plate from (−20, −30), padded 4, with a mirrored and patterned boss. Reference drawing (PNG) How: On z = 4, pad a 5 × 5 rectangle from (5, −20) by 6. In one MultiTransform, mirror the boss about YZ, then pattern along +Y: 2 occurrences, pitch 10. Set total pattern length to (Occurrences − 1) × 10.	b: pattern occurrences 3

	ID	MAKE	TWINS
☐	**N18**	A result Body built on a hidden SubShapeBinder of a visible source Body's top face. Count both Bodies: two solids, total volume 1100. Reference drawing (PNG) How: Pad a source rectangle 20 × 10 from (0, 0) by 4. In a second Body, bind its top face, sketch a 10 × 6 rectangle from (0, 0) on the binder, and pad 5. Keep both Bodies visible. Hide only the binder, and exclude only the binder from measurements.	b: source Pad length 8
☐	**N30**	A 12 × 8 rectangle starts at (4, 0), padded 2. Reference drawing (PNG) How: Use Coincident corners, Horizontal, Vertical, Parallel, and Perpendicular sides, PointOnObject at y = 0, a point-point Distance of 12, and a point-line Distance of 8. Set the starting DistanceX to 4.	b: starting DistanceX −4
☐	**N31**	A 6 × 3 rectangle starts at (−5, −4), padded 2. Reference drawing (PNG) How: Constrain a reference point at (3, 2). From that point to the lower-left corner, set DistanceX −8 and DistanceY −6.	b: DistanceX +8 · c: DistanceY +6
☐	**N32**	A right triangle has a hypotenuse of 10 at +30° from +X, padded 2. Reference drawing (PNG) How: Start the hypotenuse at the origin. Constrain its directed line-line Angle from the X axis. Close with a vertical leg and a horizontal leg on the X axis.	b: Angle −30° · c: Angle +120°
☐	**N33**	A capsule has semicircle centres (0, 0) and (8, 0), radius 2, and straight sides of length 8, padded 2. Reference drawing (PNG) How: Join the arcs and lines with endpoint-endpoint Tangent constraints, and constrain the arc radii Equal. Make the top line edge-edge Tangent to a blocked construction circle at (4, 3), radius 1. Set the right centre DistanceX to 8.	b: right centre DistanceX −8

	ID	MAKE	TWINS
☐	**N34**	A triangle has base endpoints (−2, 1) and (6, 1), and apex (2, 7), padded 2. Reference drawing (PNG) How: Constrain the base endpoints Symmetric about a vertical construction line at x = 2. Put the apex on that line, and dimension the base 8, base y = 1, and apex y = 7.	b: symmetry line DistanceX −2
☐	**N35**	An annulus has centre (−7, 0), outer radius 5, and inner diameter 4, padded 2. Reference drawing (PNG) How: Use Radius for the outer circle and Diameter for the inner circle. Add a non-driving outer Diameter and inner Radius. Keep the centres Coincident.	b: centre DistanceX +7
☐	**N36**	A circle at (−6, 2), radius 3, is blocked and padded 2 toward +Z. Reference drawing (PNG) How: Apply Block to the circle, then pad its disc. Keep the blocked sketch geometry intact for the direction twin.	b: reverse the pad toward −Z · c: replace Block with Lock where that constraint type exists
☐	**N37**	A refracted ray closes a triangle from (0, 0) to (8/√15, −8) to (0, −8), padded 2. Reference drawing (PNG) How: Block a horizontal interface and an incident ray arriving at the origin at 30° from the downward normal. Apply SnellsLaw with ratio +2, and constrain the refracted endpoint y = −8.	b: SnellsLaw ratio −2
☐	**N38**	A rational quadratic B-spline bounds a quarter-disc of radius 4, padded 2. Reference drawing (PNG) How: Use poles (4, 0), (4, 4), and (0, 4), endpoint weights 1, and middle Weight 1/√2. Expose the pole circles and knot points, constrain the middle and last poles horizontally, and close to the origin with two lines.	b: middle pole DistanceY −4

	ID	MAKE	TWINS
☐	**N39**	An ellipse has major diameter 12 from the origin at +30°, minor diameter 6, and pad length 2. Reference drawing (PNG) How: Align two construction lines to the ellipse major and minor diameters with InternalAlignment. Fix the major line start at the origin, its length 12, its directed Angle +30°, and the minor line length 6.	b: major line Angle −30°
☐	**N40**	An 8 × 6 rectangle has centre (4, 3), padded 2. Reference drawing (PNG) How: Constrain opposite corners Symmetric about a construction point at (4, 3). Keep the sides horizontal and vertical, and dimension the width 8 and height 6.	b: symmetry point DistanceY −3

Drawings

A drawing names the part it shows, so keep the two in the same folder. Send each drawing with its sheets exported as a DXF, and as a PDF too if you like: they show us what it should look like. Export the DXF at scale 1:1, in millimetres, with text kept as text (not exploded), as AutoCAD 2000 or 2018 DXF. If the app writes one DXF per sheet, zip them into one file (`D02a.zip`). Fusion makes D01 to D03, Inventor and SolidWorks make D04 and D05, and Inventor also makes D06 and D07.

Fusion keeps drawings online, so send three files for each: the drawing itself as a Fusion Archive (`D01a.f3z`, which carries its design), and its sheets exported as a DXF, plus a PDF if you like (`D01a.pdf`, `D01a.dxf`). Get the archive from the web: in the Data Panel, Open on the Web, open the drawing, Export → Fusion Archive, and save the download. Get the DXF (and a PDF, if you like) from the drawing's Export panel, all sheets. Inventor and SolidWorks: D04 is a small bracket, and D05 its drawing, sent with a DXF of the sheet (and a PDF, if you like).

	ID	MAKE	TWINS
☐	**D01**	Fusion only · High priority A drawing of F01a on one A4 landscape sheet, with the default title block as it comes: one base view, Front, scale 1:1, and nothing else. How: Open F01a, then File → New Drawing → From Design: Standard ISO, Units mm, Sheet Size A4 (landscape), the default title block. Place one Base View, Orientation Front, Scale 1:1, and click to finish without projecting any views. For each twin, open D01a, change only what it says, and save it as the twin; Fusion saves it as a new drawing in the cloud.	b: add two Projected Views from the base view: Top (above or below it, as Fusion places it) and Right (beside it) · c: add one isometric Projected View (drag diagonally from the base view) · d: Sheet Size A3, portrait (if Fusion has no portrait A3, use A3 and write that on the card) · e: add a Detail View: centre on the base view's top-right corner, circle 5 mm across, scale 2:1 · f: add two linear dimensions on the base view: 20 (the width) and 10 (the height) · g: add one note, TEST NOTE, with a leader to the base view's top edge · h: fill the title block: title D01H, part number P-001, designer VOL
☐	**D02**	Fusion only · High priority D01a + a second sheet holding only a Top base view, scale 1:1. How: Open D01a and save it as D02a. Add a sheet (the + beside the sheet tab), then a Base View on it, Orientation Top, Scale 1:1, no projected views.	none

	ID	MAKE	TWINS
☐	**D03**	Fusion only High priority A drawing of F04a (the block with the hole) on one A4 landscape sheet, default title block: a Front base view, scale 1:1, and a section view A-A through the hole's centre. How: Open F04a, then File → New Drawing → From Design: Standard ISO, Units mm, Sheet Size A4, the default title block. Base View, Orientation Front, Scale 1:1. Section View: pick the base view, draw the section line straight down through the hole's centre (x = 0), finish the line, and place the view beside it; keep its label A-A.	b: add a dimension across the hole in the section view: pick the hole's two edges, so it reads 5 (Fusion may add Ø) · c: from **b**: add a Centerline along the hole's axis in section A-A (Geometry → Centerline, pick the hole's two edges in that view)
☐	**D04**	Inventor and SolidWorks only An L-shaped bracket: on XZ, the profile (0, 0) → (40, 0) → (40, 5) → (5, 5) → (5, 30) → (0, 30) and back, extruded 30 so it spans y 0…30. Then a hole Ø6 through the base at (25, 15). It spans x 0…40, y 0…30, z 0…30. How: New part, units mm. Sketch on XZ: the L with the Line tool, dimensioned from the origin: 40, 5, 30, and 5. Finish the sketch. Extrude 30, New Body; flip the direction if it goes toward −Y, so it spans y 0…30. Hole tool (not an extrude cut): a point on the base's top face (z = 5) dimensioned to (25, 15), Simple, Ø6, Through All. Check the extents above before saving.	none

	ID	MAKE	TWINS
☐	**D05**	Inventor and SolidWorks only A drawing of D04a on one sheet, scale 1:1, with the default template and title block: a base view showing the L, two views projected from it (one above or below, one beside), and four linear dimensions: 40 (the base length), 30 (the height), 5 (the base thickness), and 25 (the hole centre from the back face at x = 0). Saved beside D04a, with its sheet exported as a DXF (and a PDF, if you like). How: New, the metric drawing template your install offers (ISO.idw or Standard.idw), unchanged. Base View of D04a, the orientation that shows the L, scale 1:1, then Projected View for the other two. Dimension the four values. Save as D05a.idw in the folder with D04a.ipt. Save Copy As, type DXF, as D05a.dxf: in its Options, AutoCAD 2018 (or 2000) DXF, scale 1:1, text not exploded.	none
☐	**D06**	High priority Inventor only A plate with one sloped corner and a hole: on XY, the profile (0, 0) → (60, 0) → (60, 20) → (40, 40) → (0, 40) and back, extruded 10 up, then a hole Ø10 through at (20, 20). It spans x 0…60, y 0…40, z 0…10. How: New part, units mm. Sketch on XY: the five lines, dimensioned from the origin: 60, 20, 40, and 40 (the sloped line runs from (60, 20) to (40, 40)). Finish the sketch. Extrude 10 up, New Solid. Hole command: a point on the top face dimensioned to (20, 20), Simple, Ø10, Through All. Check the extents above before saving.	none

	ID	MAKE	TWINS
☐	**D07**	High priority Inventor only A drawing of D06a on two sheets, with the default title block on each. Sheet 1: A4, portrait. Sheet 2: Letter, landscape. On each sheet: a base view of D06a at scale 1:1, a projected view above or below it at 1:2, and a projected view beside it at 2:1, with every view label shown. On sheet 1's base view, four dimensions: linear 60 (the bottom edge), aligned 28.28 (the sloped edge), radial R5 (the hole), and angular (between the top edge and the sloped edge). How: New, the metric drawing template your install offers (ISO.idw or Standard.idw). Right-click the sheet in the browser, Edit Sheet: Size A4, Portrait. Base View of D06a, the orientation that shows the sloped corner and the hole (Front in a new part), Scale 1:1, Show Label on. Projected View: one above or below and one beside. Edit each projected view: untick Style from Base, set its Scale (1:2 for the one above or below, 2:1 for the one beside), and Show Label on. Dimension on the base view: Dimension on the bottom edge (60), Dimension on the sloped edge choosing Aligned (28.28), Dimension on the hole's circle choosing Radius (R5), and Dimension clicking the top edge and then the sloped edge (an angle). New Sheet: Edit Sheet, Size A (Letter), Landscape, and place the same three views at the same scales, labels on. Save as D07a.idw in the folder with D06a.ipt. Save Copy As, type DXF, as D07a.dxf, all sheets (Inventor writes one DXF per sheet, so zip them as D07a.zip): Options AutoCAD 2018 (or 2000) DXF, scale 1:1, text not exploded. In the upload note, write the four dimension values as Inventor shows them and, if your Inventor shows them, each view's centre on its sheet in mm.	none

	ID	MAKE	TWINS
☐	**D08**	High priority Fusion only A drawing of the F10a assembly (block and pin) on one A4 landscape sheet, default title block: a Front base view of the whole assembly, scale 1:1, and a parts list. How: Open F10a, then File → New Drawing → From Design: Standard ISO, Units mm, Sheet Size A4, the default title block, the whole assembly. Base View, Orientation Front, Scale 1:1. Then Tables → Table (the parts list), click the base view, and place it at the top right.	b: from **a**: add a Balloon to each of the two parts in the base view
☐	**D09**	High priority Fusion only A drawing made before its part changed, and the same drawing updated after the change. How: Open F01a and Save As D09-part. Make a drawing of D09-part exactly as D01a (one Front base view, scale 1:1, A4 landscape, default title block) and save it as D09a. Close it.	b: open D09-part, edit its Extrude from 10 to 12, and save. Then open D09a: Fusion shows the drawing is out of date. Update it to the new version, then Save As D09b

CAM

These recipes are for Fusion's Manufacture workspace only (M IDs), and they are high priority: Fusion's free plan is changing, and 3D toolpaths may soon leave it, so we'd like them while they can still be made. Each one is a small part with a setup and one kind of operation, with every value given, so we can check what we read against the recipe. Units mm. Leave every value the recipe doesn't name at Fusion's default. Make the parts in the Design workspace first, then switch to Manufacture, Milling tab. Generate every operation (it shows a green tick) before you save. Save each file as its ID (`M03a`) and export it with *File → Export → .f3d*; an *.f3z* is fine too. Every operation uses the three tools of M02, picked from your *CAM fixtures* library, with spindle speed 10000 rpm and the feeds the recipe gives. Two parts are used: the **CAM plate** (M01) and the **CAM dome** (M07).

	ID	MAKE	TWINS
☐	**M01** r2	Fusion only High priority The CAM plate and one setup. The plate: a 60 (X) × 40 (Y) rectangle centred on the origin on XY, extruded 15 up; a 30 × 20 pocket centred on the top face, corners R5, 5 deep; and four Ø6 holes through at (±22, ±14). It spans x −30…30, y −20…20, z 0…15. The setup: Milling, the WCS in model orientation with its origin at the top centre of the stock, and a stock box 2 larger on each side and 1 above the part. Reference drawing (PNG) How: Design: Sketch on XY, Center Rectangle from the origin, 60 along X and 40 along Y. Extrude 15 up, New Body. Sketch on the top face: Center Rectangle 30 × 20 from the origin, Sketch Fillet R5 on its four corners; Extrude −5, Cut. Hole tool on the top face, four points at (22, 14), (−22, 14), (−22, −14), and (22, −14), Simple, Ø6, Through All. Manufacture: Setup → New Setup. Setup tab: Operation Type Milling; Work Coordinate System: Orientation *Model orientation*, Origin *Stock box point*, and click the point at the top centre of the stock box. Stock tab: Mode *Relative size box*, Stock Offset Mode *Add stock to sides and top-bottom*, Stock Side Offset 2, Stock Top Offset 1, Stock Bottom Offset 0. OK, and leave the setup named Setup1.	b: from **a**: Stock Top Offset 2 · c: from **a**: WCS Origin at the stock box point on the top face's front-left corner (x −32, y −22, z 16) · d: from **a**: WCS Origin at the top face's back-right corner (x 32, y 22, z 16) · e: from **a**: WCS Origin at the bottom face's front-left corner (x −32, y −22, z 0) · f: from **a**: WCS Origin at the top face's front-right corner (x 32, y −22, z 16)

	ID	MAKE	TWINS
☐	**M02** r2	Fusion only High priority A tool library of three tools, and M01a with one Face operation that uses tool 1. Send the library too, exported as `M02a.tools`. Tool 1: flat end mill Ø6, 3 flutes. Tool 2: ball end mill Ø6, 2 flutes. Tool 3: drill Ø6, tip angle 118°. How: Manufacture → Manage → Tool Library. Under Local, right-click Library → New library, and name it *CAM fixtures*. In it, add three tools, units mm, with these Cutter values and the Post Processor Number shown. Tool 1, Flat end mill: Diameter 6, Number of flutes 3, Flute length 15, Shoulder length 20, Overall length 50, Shaft diameter 6, Length below holder 25; Number 1. Tool 2, Ball end mill: Diameter 6, Number of flutes 2, the same lengths; Number 2. Tool 3, Drill: Diameter 6, Tip angle 118°, Flute length 30, Overall length 60, Shaft diameter 6, Length below holder 40; Number 3. Right-click the library, Export, and save it as M02a.tools. Then open M01a, save it as M02a, and add 2D → Face: Tool 1, Spindle Speed 10000 rpm, Cutting Feedrate 1000 mm/min; Passes: Stepover 4.5; everything else as it comes.	b: from **a**: Face Stepover 3
☐	**M03** r2	Fusion only High priority M02a + a 2D Adaptive Clearing of the pocket with tool 1. How: Open M02a, save it as M03a. 2D → 2D Adaptive Clearing. Tool: tool 1, Spindle Speed 10000 rpm, Cutting Feedrate 1000 mm/min. Geometry: Pocket Selections, the pocket's bottom edge loop. Passes: Optimal Load 1; Stock to Leave ticked, Radial Stock to Leave 0.3, Axial Stock to Leave 0.2; Multiple Depths ticked, Maximum Roughing Stepdown 2. OK, and let it generate.	b: from **a**: Optimal Load 1.5
☐	**M04** r2	Fusion only High priority M02a + a 2D Pocket of the pocket with tool 1. How: Open M02a, save it as M04a. 2D → 2D Pocket. Tool: tool 1, Spindle Speed 10000 rpm, Cutting Feedrate 800 mm/min. Geometry: Pocket Selections, the pocket's bottom edge loop. Passes: Maximum Stepover 2.4; Multiple Depths ticked, Maximum Roughing Stepdown 2; Stock to Leave unticked.	b: from **a**: Maximum Stepover 3

	ID	MAKE	TWINS
☐	**M05** r2	Fusion only High priority M02a + a 2D Contour round the outside of the plate with tool 1, cutting 1 below its bottom. How: Open M02a, save it as M05a. 2D → 2D Contour. Tool: tool 1, Spindle Speed 10000 rpm, Cutting Feedrate 1000 mm/min. Geometry: Contour Selections, the plate's bottom outer edge loop; Tabs unticked. Heights: Bottom Height *Selected contour(s)*, Offset −1. Passes: Multiple Depths ticked, Maximum Roughing Stepdown 3; Stock to Leave unticked.	b: from **a**: Tabs ticked, Tab Shape Rectangular, Tab Width 5, Tab Height 2, Tab Positioning By distance, Tab Distance 50
☐	**M06** r2	Fusion only High priority M02a + Drilling of the four Ø6 holes with tool 3. How: Open M02a, save it as M06a. Drilling → Drilling. Tool: tool 3, Spindle Speed 10000 rpm, Plunge Feedrate 200 mm/min. Geometry: Hole Mode *Selected faces*, and pick the four holes' walls. Heights: Bottom Height *Hole bottom*; Drill Tip Through Bottom ticked, Break-Through Depth 1. Cycle: Cycle Type *Drilling - rapid out*.	b: from **a**: Cycle Type *Deep drilling - full retract*, Pecking Depth 3
☐	**M07** r2	Fusion only High priority The CAM dome, a setup like M01's, and a 3D Adaptive Clearing with tool 1. The dome: a 60 (X) × 40 (Y) rectangle centred on the origin on XY, extruded 20 up, with R10 fillets on its four top edges. It spans x −30…30, y −20…20, z 0…20. Reference drawing (PNG) How: New design. Sketch on XY, Center Rectangle from the origin, 60 along X and 40 along Y. Extrude 20 up, New Body. Fillet the four top edges in one Fillet, R10. Manufacture: a setup exactly as in M01 (Milling; Model orientation; origin at the stock box's top centre; Relative size box, side offset 2, top offset 1, bottom offset 0). Then 3D → Adaptive Clearing. Tool: tool 1 from CAM fixtures, Spindle Speed 10000 rpm, Cutting Feedrate 1000 mm/min. Passes: Optimal Load 1, Maximum Stepdown 2; Stock to Leave ticked, Radial Stock to Leave 0.5, Axial Stock to Leave 0.5.	b: from **a**: Maximum Stepdown 3

	ID	MAKE	TWINS
☐	**M08** r2	Fusion only High priority M07a + a Parallel finish with tool 2. How: Open M07a, save it as M08a. 3D → Parallel. Tool: tool 2, Spindle Speed 10000 rpm, Cutting Feedrate 1200 mm/min. Passes: Stepover 0.5, Pass Direction 0°; Stock to Leave unticked.	b: from **a**: Pass Direction 90°
☐	**M09** r2	Fusion only High priority M07a + a Scallop finish with tool 2. How: Open M07a, save it as M09a. 3D → Scallop. Tool: tool 2, Spindle Speed 10000 rpm, Cutting Feedrate 1200 mm/min. Passes: Stepover 0.3; Stock to Leave unticked.	b: from **a**: Stepover 0.5
☐	**M10** r2	Fusion only High priority Two setups: M04a, plus a second setup for the flipped part with a Face of the underside. How: Open M04a, save it as M10a. Setup → New Setup. Setup tab: Operation Type Milling; Work Coordinate System: Orientation *Select Z axis/plane & X axis*, pick the plate's bottom face for Z and tick Flip Z Axis so Z points down, out of the bottom; Origin *Stock box point*, the point at the centre of the stock's bottom face. Stock tab: as in M01. OK, so it is named Setup2. Then, with Setup2 selected, 2D → Face: tool 1, Spindle Speed 10000 rpm, Cutting Feedrate 1000 mm/min, Stepover 4.5.	none
☐	**M11**	Fusion only High priority M05a with a post processor chosen in the setup, and its NC program. Send the NC file too if Fusion lets you post (`M11a.nc`); it's welcome but optional. How: Open M05a, save it as M11a. Edit Setup1, Post Process tab: Program Name/Number 1001, Program Comment M11. Then right-click Setup1 → Create NC Program: Post *Fanuc / fanuc* from the Fusion library, Program name 1001, Program comment M11, Units Millimeters, Output folder anywhere. OK, so an NC Program appears in the browser and is saved with the design. Right-click it → Post Process to write M11a.nc (some free plans can't post; then send just the design).	b: from **a**: Post *Haas / haas* instead

NX

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs, except those marked Inventor only and A09) with the NX tools in this table. Start from the Model template in millimetres, then switch to the NX Sheet Metal application for S recipes. NX is Z up: keep the recipe's coordinates and positive directions as written. Save parts and assemblies alike as `.prt`, for example `P01a.prt`, `S02b.prt`, and `A02a.prt`. Keep the component files beside each assembly. Choose your release in the portal, for example NX 2412; each release is its own set, made and saved in that release. Fully constrain sketches, and make every twin from its a file with just the one change. Use Arrangements for A08's component suppression. Leave out A09: NX has no plain equivalent to an Inventor named view representation that stores component visibility and is saved active. If another option differs, note what you used on the card. Check volume with Analysis, Measure Bodies. Files are checked by hand.

RECIPE SAYS	IN NX
Planes XY · XZ · YZ (coordinates as written)	Datum CSYS planes XY · XZ · YZ; NX is Z up. Keep x, y, z, and their positive directions as written. Sketch on the named plane, with dimensional and geometric constraints
Extrude (P01, P07–P09, P12, P13, P16)	Extrude; use a circle for the discs and pin, and a sketch on the end face for the boss. Set the Boolean option to Unite when adding to the same body
Revolve (P02)	Revolve, 360° about the sketch axis; keep the ring above XY
Hole command (P03, P07, P08)	Hole, Simple, Through Body, positioned with constrained sketch points (P03b: Counterbored, Ø9, depth 3)
Fillet (P04) · Chamfer (P05) · Shell (P06)	Edge Blend, constant radius · Chamfer, 45° · Shell, remove the top face, inside thickness
Rectangular / Circular Pattern (P07, P08) · Mirror (P09)	Pattern Feature, Linear, counts and pitch in both directions · Pattern Feature, Circular about Z, equally spaced over 360° · Mirror Feature of the boss about XZ
Sweep (P10)	Sweep Along Guide with the circle as section and the rounded sketch path as guide, or Swept with one section and one guide. Use a sketch fillet for the path's R10. Datum Plane, normal to curve at the path start, for the circle; note the orientation option used

RECIPE SAYS	IN NX
Loft (P11)	Through Curves between the square and circle sketches, with a solid output; align the section start points and directions, and note the alignment option
User parameter plate_w (P12)	Expressions: create a length expression `plate_w = 20` in mm; use `plate_w` for sketch width and `plate_w / 2` for Extrude distance
Work plane offset (P11, P13)	Datum Plane, offset from the XY datum plane; sketch on the resulting plane
3D sketch (P14)	Lines through the three coordinate points in space, joined as a guide; Sweep Along Guide a Ø3 circle on a Datum Plane normal to the guide at its start. For b, change only the last point's z coordinate
Appearance (P15, A10)	Edit Object Display, Color, on the body or face; select the component occurrence for A10's assembly color override
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	NX Sheet Metal application → Sheet Metal Preferences: thickness 2, bend radius 1, neutral factor 0.44, round relief size 4, and gap 0.5 where offered; record the settings
Face (S01) · Flange (S02, S03)	NX Sheet Metal → Tab from the rectangle · Flange, height 20, angle 90°. For S03, select both edges in one feature where supported. Contour Flange makes a wall from an open section; use Bend for a bend across an existing tab
Height datum · bend position · width (S02)	Flange: change the length reference, material side, or width limits to match the named twin, keeping the other values. Record the options used
Hem (S04)	NX Sheet Metal → Hem on the flange's free edge, toward −Y, gap 1 (b: 1.5). Use Jog only for a recipe that calls for an offset wall, recording its height and bend settings
Corner relief / gap (S03) · Bend relief (S06)	Flange and Sheet Metal Preferences: set the named corner or bend relief shape, size, width, or gap; record the options used

RECIPE SAYS	IN NX
Flat Pattern, back to folded (S05)	NX Sheet Metal → Flat Pattern for the flat dimensions; save with the folded body displayed. Unbend can unfold selected bends; Rebend restores them before saving
Corner Round (S07)	NX Sheet Metal → Break Corner, round, radius 3; c: select all four named corners in the same feature
Place parts · Ground / unground (A01)	Add Component to insert P01a and P02a · Assembly Constraints, Fix each component for a; remove the Fix constraints for b
Mate · Flush · Angle (A02–A04)	Assembly Constraints → Touch Align, opposed faces (A02b: Distance 5) · Touch Align, aligned faces · Angle; select the recipe's faces and record the orientation
Insert (A05) · Tangent (A06)	Assembly Constraints → Concentric on the axes plus Touch Align on the seating faces (A05b: Distance 2) · Touch Align with the tangent solution between cylinder and plane; record the inside or outside solution
Pattern Component (A07)	Component Pattern, Linear, direction +X, 3 instances, spacing 70 (b: 4); keep P01a in place
Model state (A08)	Arrangements: keep the default arrangement with both components, and create a second arrangement with P02a suppressed only in that arrangement. Save with the default active for a, and the second active for b

Solid Edge

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs, except those marked Inventor only) with the Solid Edge tools in this table. Start from part, sheet-metal, and assembly templates in millimetres. Use the ordered (history) modeling mode, not synchronous, so features keep their order. Use Solid Edge's own Sheet Metal environment for S recipes. Save parts as `P01a.par`, sheet-metal parts as `S01a.psm`, and assemblies as `A01a.asm`. Choose your year in the portal, for example Solid Edge 2026; each year is its own set, made and saved in that year. Use Alternate Assemblies for A08 and Display Configurations for A09. Check volume with Inspect, Physical Properties. Fully constrain sketches, keep the written coordinates and positive directions, and make every

twin from its a file with just the one change. Keep component files beside each assembly. If an option differs, note what you used on the card. Files are checked by hand.

RECIPE SAYS	IN SOLID EDGE
Planes XY · XZ · YZ (coordinates as written)	Reference planes XY · XZ · YZ; sketch on the named plane with dimensional and geometric relationships. Keep x, y, z, and their positive directions as written
Extrude (P01, P07–P09, P12, P13, P16)	Protrusion in Ordered mode; use a circle for the discs and pin, and a sketch on the end face for the boss
Revolve (P02)	Revolved Protrusion, 360° about the sketch axis; keep the ring above XY
Hole command (P03, P07, P08)	Hole, simple, Through All, with constrained hole centers (P03b: Counterbore, Ø9, depth 3)
Fillet (P04) · Chamfer (P05) · Shell (P06)	Round, constant radius · Chamfer, 45° · Thin Wall, remove the top face, thickness inward
Rectangular / Circular Pattern (P07, P08) · Mirror (P09)	Rectangular Pattern of the hole, counts and pitch in both directions · Circular Pattern about Z, equally spaced over 360° · Mirror Copy Feature of the boss about XZ
Sweep (P10)	Swept Protrusion, circle section along the rounded sketch path. Round the path corner R10; use a reference plane normal to the path at its start for the circle, and record the orientation option
Loft (P11)	Lofted Protrusion between the square and circle sketches; align section start points and directions, and note the mapping option
User parameter plate_w (P12)	Variable Table: create a distance variable `plate_w = 20` in mm; set sketch width to `plate_w` and Protrusion distance to `plate_w / 2`
Work plane offset (P11, P13)	Reference Plane, Parallel Plane, offset from XY; sketch on the resulting plane

RECIPE SAYS	IN SOLID EDGE
3D sketch (P14)	3D Sketch, lines through the three coordinate points; Swept Protrusion of a Ø3 circle on a reference plane normal to the path at its start. For b, change only the last point's z coordinate
Appearance (P15, A10)	Part Painter: part red; b: top face blue. In the assembly, apply a blue face style to the component occurrence for A10
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	Sheet Metal environment, Material Table and bend options: thickness 2, inside bend radius 1, neutral factor 0.44, round relief size 4, and gap 0.5 where offered; record the settings
Face (S01) · Flange (S02, S03)	Tab from the rectangle · Flange, height 20, angle 90°. For S03, select both edges in one feature where supported. Contour Flange makes a wall from an open section
Height datum · bend position · width (S02)	Flange options: change the height reference, material side, or width limits to match the named twin; keep the other values, and record the options used
Hem (S04)	Hem on the flange's free edge, toward –Y, gap 1 (b: 1.5); record the hem type
Corner relief / gap (S03) · Bend relief (S06)	Flange and bend options: set the named corner or bend relief shape, size, width, or gap; record the options used
Flat Pattern, back to folded (S05)	Flat Pattern for the flat dimensions; return to the folded model before saving
Corner Round (S07)	Break Corner, round, radius 3; c: select all four named corners in the same feature
Place parts · Ground / unground (A01)	Place Part to insert P01a and P02a · Ground both occurrences for a; remove the Ground relationships for b
Mate · Flush · Angle (A02–A04)	Assembly relationships: Mate, opposed faces (A02b: offset 5) · Planar Align, aligned faces · Angle; select the recipe's faces, and record the orientation

RECIPE SAYS	IN SOLID EDGE
Insert (A05) · Tangent (A06)	Axial Align on the axes plus Mate on the seating faces for Insert (A05b: offset 2) · Tangent, with the named inside or outside solution
Pattern Component (A07)	Pattern Parts, rectangular, direction +X, 3 instances, spacing 70 (b: 4); keep P01a in place
Model state (A08)	Alternate Assemblies: make an alternate member with P02a excluded; keep both parts in the primary member. Save the primary active for a, and the alternate active for b
View representation (A09)	Display Configurations: save a configuration named View1 with P02a hidden; retain a default configuration with both visible. Save the default active for a, and View1 active for b

Alibre Design

Make the Inventor part, sheet-metal, and assembly recipes above (P, S, and A IDs, except those marked Inventor only and A09) with the Alibre Design tools in this table. Start from part, sheet-metal, and assembly templates in millimetres. Save parts as `P01a.AD_PRT`, sheet-metal parts as `S01a.AD_SMP`, and assemblies as `A01a.AD_ASM`. Choose your release in the portal, for example Alibre Design 28; each release is its own set, made and saved in that release. Use Configurations for A08; sheet metal and configurations need an edition that includes them. Leave out A09: Alibre has no plain equivalent to a named view representation that stores component visibility and is saved active. Check volume with Tools, Physical Properties. Fully constrain sketches, keep the written coordinates and positive directions, and make every twin from its a file with just the one change. Keep component files beside each assembly. If an option differs, note what you used on the card. Files are checked by hand.

RECIPE SAYS	IN ALIBRE DESIGN
Planes XY · XZ · YZ (coordinates as written)	XY-Plane · XZ-Plane · YZ-Plane; sketch on the named plane with dimensional and geometric constraints. Keep x, y, z, and their positive directions as written
Extrude (P01, P07–P09, P12, P13, P16)	Extrude Boss; use a circle for the discs and pin, and a sketch on the end face for the boss

RECIPE SAYS	IN ALIBRE DESIGN
Revolve (P02)	Revolve Boss, 360° about the sketch axis; keep the ring above XY
Hole command (P03, P07, P08)	Hole, simple, Through All, with a constrained placement sketch (P03b: Counterbore, Ø9, depth 3)
Fillet (P04) · Chamfer (P05) · Shell (P06)	Fillet, constant radius · Chamfer, 45° · Shell, remove the top face, thickness inward
Rectangular / Circular Pattern (P07, P08) · Mirror (P09)	Linear Feature Pattern of the hole, counts and pitch in both directions · Circular Feature Pattern about Z, equally spaced over 360° · Mirror of the boss about XZ
Sweep (P10)	Sweep Boss, circle section along the rounded sketch path. Sketch Fillet R10 for the path corner; insert a plane normal to the path at its start for the circle, and record the orientation option
Loft (P11)	Loft Boss between the square and circle sketches; align section start points and directions, and note the connection option
User parameter plate_w (P12)	Equation Editor: create a distance parameter `plate_w = 20` in mm; set sketch width to `plate_w` and Extrude Boss depth to `plate_w / 2`
Work plane offset (P11, P13)	Insert Plane, offset from XY-Plane; sketch on the resulting plane
3D sketch (P14)	3D Sketch, lines through the three coordinate points; Sweep Boss of a Ø3 circle on a plane normal to the path at its start. For b, change only the last point’s z coordinate
Appearance (P15, A10)	Color Properties: part red; b: top face blue. In the assembly, set the component instance color blue for A10
Sheet Metal Defaults (thickness 2, bend R 1, K 0.44, relief round 4, gap 0.5)	Sheet Metal Parameters: thickness 2, inside bend radius 1, K factor 0.44, round relief size 4, and gap 0.5 where offered; record the settings

RECIPE SAYS	IN ALIBRE DESIGN
Face (S01) · Flange (S02, S03)	Tab from the rectangle · Flange, height 20, angle 90°. For S03, select both edges in one feature where supported. Contour Flange makes a wall from an open section
Height datum · bend position · width (S02)	Flange options: change the length reference, bend position, or width limits to match the named twin; keep the other values, and record the options used
Hem (S04)	Hem on the flange's free edge, toward –Y, gap 1 (b: 1.5); record the hem type
Corner relief / gap (S03) · Bend relief (S06)	Flange and Sheet Metal Parameters relief options: set the named corner or bend relief shape, size, width, or gap; record the options used
Flat Pattern, back to folded (S05)	Flat Pattern for the flat dimensions; return to the folded model before saving
Corner Round (S07)	Corner Round, radius 3; c: select all four named corners in the same feature
Place parts · Ground / unground (A01)	Insert the P01a and P02a parts · Anchor both for a; remove their anchors for b
Mate · Flush · Angle (A02–A04)	Assembly constraints: Mate, opposed faces (A02b: offset 5) · Align, aligned faces · Angle; select the recipe's faces, and record the orientation
Insert (A05) · Tangent (A06)	Insert constraint, or Align axes plus Mate seating faces (A05b: offset 2) · Tangent, with the named inside or outside solution
Pattern Component (A07)	Linear Component Pattern, direction +X, 3 instances, spacing 70 (b: 4); keep P01a in place
Model state (A08)	Configurations: create a second configuration with P02a suppressed, locking suppression separately from the original. Save the original active for a, and the second active for b

Onshape

These recipes are for Onshape only (O IDs). Make each one as its own new document, named by its ID (`001a`), with everything in the one Part Studio it starts with. Set the workspace units to millimetres before you start, and use the default planes: Top is the XY plane, Front is XZ, and Right is YZ. Dimension everything from the origin, fully define every sketch, and use exactly the feature each recipe names. Onshape runs in the browser and updates itself, so there is no version to pick in the portal.

> **Onshape: how to send a file.** Onshape keeps documents online, so each one is sent as an `.onshape-bundle` file that Dutchman makes from it. In Dutchman, use the Onshape option, paste the link to the document's Part Studio, and open it. Dutchman saves the bundle beside the design it makes, and lets you show the bundle or send it straight to us. Otherwise, upload that bundle on the recipe's page in the portal. The first time, Dutchman asks for your Onshape API keys: they stay on your computer, so never upload them or paste them anywhere else.

	ID	MAKE	TWINS
☐	**001**	Onshape only One sketch on Top and no features after it: a 60 × 40 rectangle centred on the origin, a Ø10 circle at (15, 10), a 3-point arc from (−20, −10) to (−10, −10) through (−15, −5), and a spline through (−25, 5), (−15, 15), (−5, 5), and (5, 15). How: Sketch on Top. Center point rectangle from the origin, Dimension 60 along X and 40 along Y; keep its horizontal and vertical constraints. Circle, centre dimensioned 15 and 10 from the origin, Dimension the diameter 10. 3 point arc: its two ends at (−20, −10) and (−10, −10), then the point it passes through at (−15, −5), each dimensioned from the origin. Spline through the four points in order, each point dimensioned from the origin. Close the sketch and add nothing else.	none
☐	**002**	Onshape only A 60 × 40 rectangle centred on Top, extruded 10 up; a Ø10 hole 5 deep at (15, 10) on the top face; a Ø6 hole through at (−15, −10). How: Sketch on Top, Center point rectangle from the origin, 60 along X and 40 along Y. Extrude, New, Blind, 10 up (+Z). Sketch on the top face (z = 10), a Ø10 circle at (15, 10). Extrude, Remove, Blind, 5 down into the part. Sketch on the top face again, a Ø6 circle at (−15, −10). Extrude, Remove, Through all.	none

	ID	MAKE	TWINS
☐	**003**	Onshape only A 40 × 40 × 20 block at x −20…20, y 20…60, z 0…20; a 10 × 10 peg from the Front plane up to the block's near face; then a Ø6 hole cut up from Top to the next face. How: Sketch on Top, Corner rectangle from (−20, 20) to (20, 60), dimensioned from the origin. Extrude, New, Blind, 20 up. Sketch on Front, a 10 × 10 rectangle x −5…5, z 5…15, dimensioned from the origin. Extrude, Add, toward the block (+Y; flip the arrow if it points away), end type Up to face, and pick the block's face at y = 20. The peg spans y 0…20. Sketch on Top, a Ø6 circle at (0, 40). Extrude, Remove, upward (+Z), end type Up to next.	none
☐	**004**	Onshape only A full revolve of an L profile about the Z axis, then a second profile revolved 90° as a new part. How: Sketch on Front: a construction line on the vertical axis through the origin, and a closed L profile 20 wide, 30 tall, and 5 thick: a foot x 10…30, z 0…5 and an upright x 10…15, z 0…30, dimensioned from the origin. Revolve, New, Full, about the construction line. Sketch on Front again: a construction line on the vertical axis through the origin and a rectangle x 40…50, z 0…10. Revolve, New, One direction, 90°, about that line. Either direction is fine.	none
☐	**005**	Onshape only A 50 × 30 × 20 block, R3 fillets on its four vertical edges in one feature, an R2 fillet on one top edge that runs round the top, and a 2 chamfer on two bottom edges. How: Sketch on Top, Center point rectangle from the origin, 50 along X and 30 along Y. Extrude, New, Blind, 20 up. Fillet, radius 3, the four vertical edges in one feature. Fillet, radius 2, the top edge at y = +15 only, with Tangent propagation on, so it follows the whole top loop. Chamfer, Equal distance 2, the bottom edges at y = −15 and y = +15; leave Tangent propagation as it comes.	none
☐	**006**	Onshape only A 60 × 40 × 15 block with four Hole features on the top face: simple Ø5 through, counterbore Ø6 with Ø10 depth 4, countersink Ø6 with Ø12 at 90°, and tapped M6 depth 12. How: Sketch on Top, Center point rectangle from the origin, 60 along X and 40 along Y. Extrude, New, Blind, 15 up. Sketch on the top face (z = 15): four points at (−20, 10), (20, 10), (−20, −10), and (20, −10), dimensioned from the origin. Then one Hole feature per point. At (−20, 10): Simple, Ø5, Through. At (20, 10): Counterbore, Ø6 through, counterbore Ø10, depth 4. At (−20, −10): Countersink, Ø6 through, countersink Ø12, angle 90°. At (20, −10): Simple, Blind, standard ISO, Tapped, size M6 × 1, hole depth 12, tapped depth 12.	none

	ID	MAKE	TWINS
☐	**007**	Onshape only Variables `#len = 80` and `#wid = #len / 2`; a rectangle dimensioned with them, extruded `#len / 8` (80 × 40 × 10). How: Variable feature, name len, value 80 mm. A second Variable feature, name wid, value #len / 2. Sketch on Top, Center point rectangle from the origin; type #len into the dimension along X and #wid into the one along Y. Extrude, New, Blind, and type #len / 8 as the depth.	none
☐	**008**	Onshape only A 40 × 40 × 40 block shelled 2 thick with its top face removed, then a separate loft from a 20 × 20 square 50 above Top to a Ø10 circle 80 above Top. How: Sketch on Top, Center point rectangle from the origin, 40 × 40. Extrude, New, Blind, 40 up. Shell, remove the top face (z = 40), thickness 2, inward. Plane, Offset from Top, 50. Plane, Offset from Top, 80. On the first plane, a 20 × 20 square centred on the origin; on the second, a Ø10 circle centred on the origin. Loft, New, with the square and the circle as its profiles.	none
☐	**009**	Onshape only An L-bracket: legs 60 and 40 long, 5 thick, 30 wide, with a Ø6 hole through each leg, an R5 fillet on the inside bend, and a 1 chamfer on the outer edge at the end of each leg. How: Sketch on Front, a closed L from the origin: a floor x 0…60, z 0…5 and a wall x 0…5, z 0…40, dimensioned from the origin. Extrude, New, Symmetric, 30, so it spans y −15…15. Sketch on the floor's top face (z = 5), a Ø6 circle at x 40, y 0; Extrude, Remove, Through all. Sketch on the wall's outer face (x = 0), a Ø6 circle at y 0, z 25; Extrude, Remove, Through all. Fillet, radius 5, the inside bend edge (x = 5, z = 5). Chamfer, Equal distance 1, the edge at the top of the wall's outer face (x = 0, z = 40) and the edge at the end of the floor's underside (x = 60, z = 0).	none

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