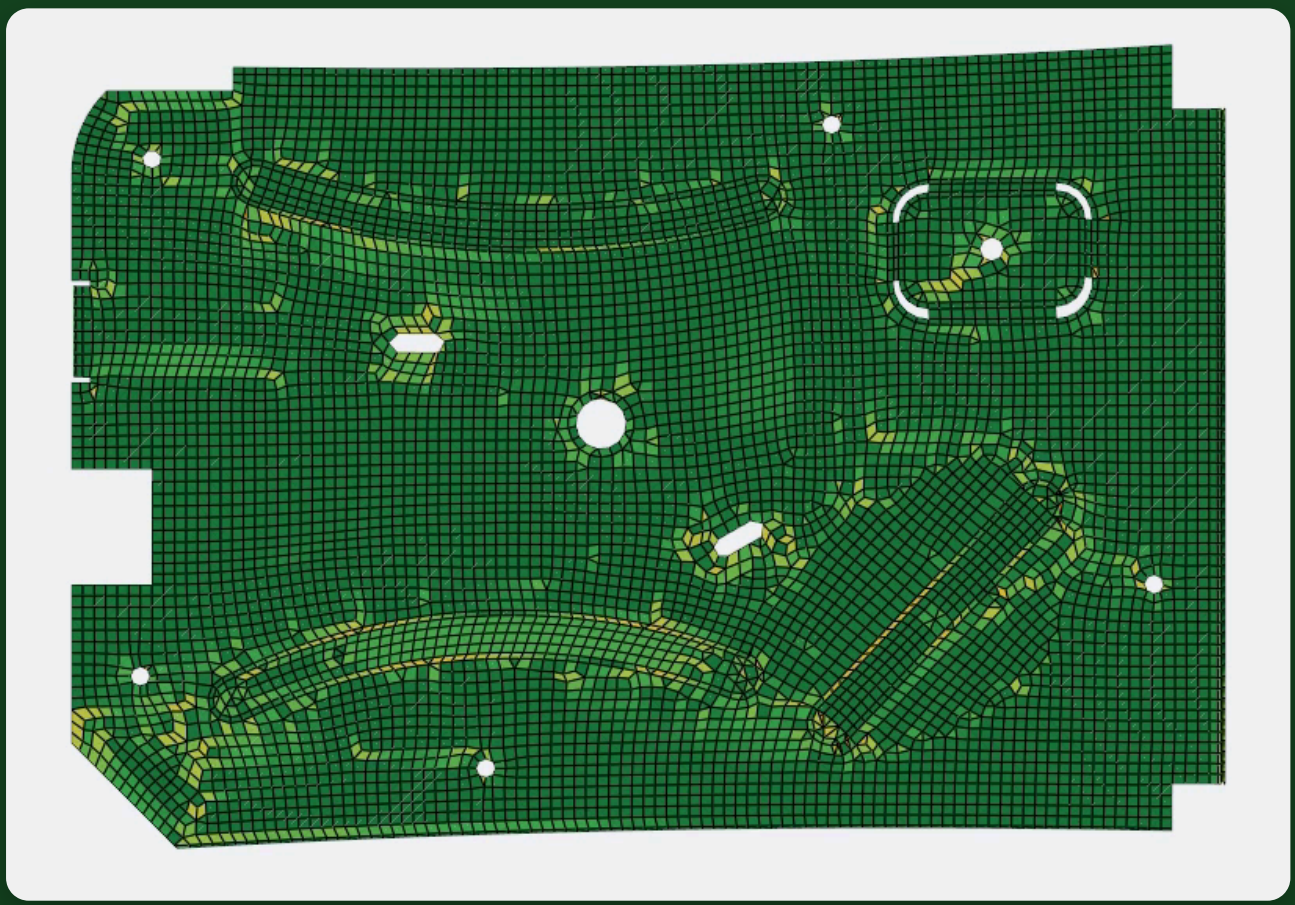
BEIDER SIMULATIONS

BeidPrep 2.0.0 beta Tutorial Workbook

Every step of the tutorial videos, with pictures



Every picture in this workbook is a frame of the tutorial videos, recorded live in BeidPrep.

Contents

One chapter per tutorial video. Open the tutorial files and follow along: each chapter names the file to open first.

Download the tutorial files (97 MB)
Download the solver results (225 MB) (videos 09 to 12)

01	Getting started: a tour of BeidPrep
02	The model tree, visibility and groups
03	The card editor
04	From CAD to shell mesh: midsurface, thickness parts, mesh check
05	Mesh editing: nodes (move, morph, create, equivalence)
06	Mesh editing: elements (combine, split, fill, normals)
07	Connectors: washers and rivets
08	Export checks and running OpenRadioss
09	From CAD to crash results: the full chain
10	Results: contour, deform, section, threshold, probe
11	Time histories: energies, contact force, peaks
12	The fibre module: short-fibre composites from tensile curves
13	SPH Generator: a bird strike in four steps
14	Working with Claude: Claude Code on the model you have open

TUTORIAL VIDEO 01

01 Getting started: a tour of BeidPrep

A first look round BeidPrep: the window, opening a model, turning and zooming, the display buttons, the difference between Save and Export, opening an OpenRadioss deck, and where to find the manual.

Tutorial files	folder `01_Getting_started` — open `sshape.beid` first
Video	Watch video 01 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, The window

video 0:00

This video takes you once around BeidPrep.

You will open a model, look round it, and see where your work is saved.

BeidPrep opens with an empty model.

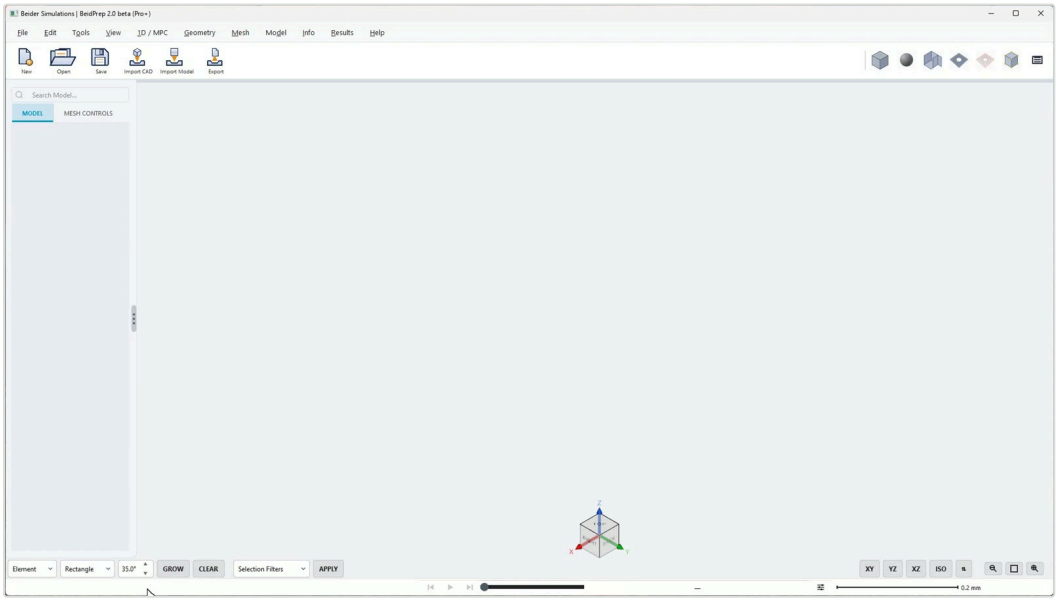
Along the top are the menus and the ribbon tabs.

A click on a tab, such as File, opens its tools in the ribbon below.

A second click closes it.

The model tree is on the left, and the 3D view fills the middle.

At the bottom, the status line says what the program just did.



The screen at 0:27 in the video.

2 Open a model

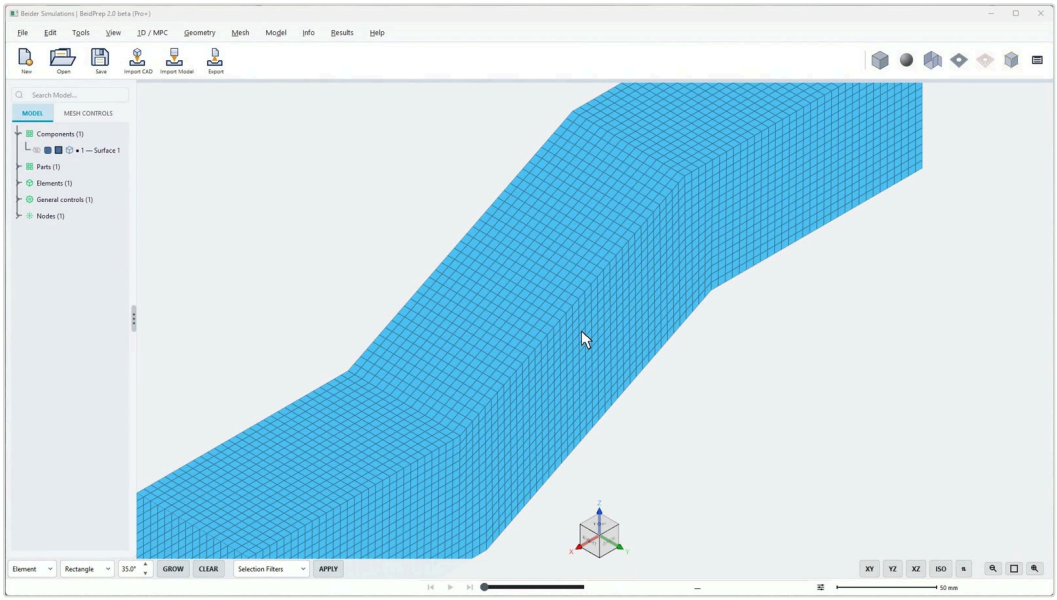
video 0:27

1. Press Control O, and choose sshape dot beid.

   The CAD, the mesh and the cards come back.

2. Hide the CAD with the eye on its row in the tree, to see the mesh.

   In the video: **sshape.beid** in open dialog: File name box.



The screen at 0:37 in the video.

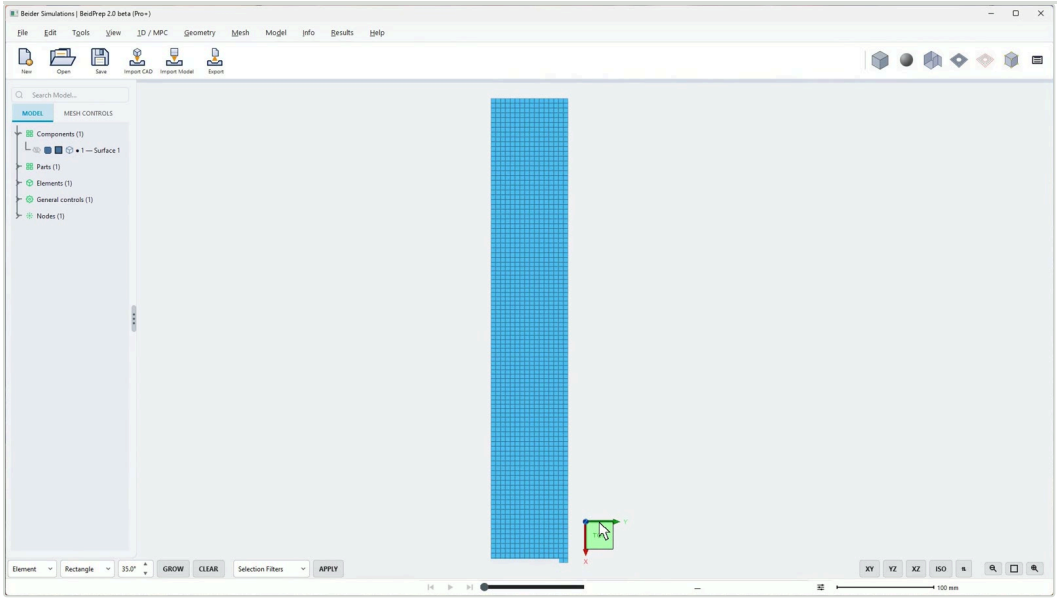
3 Turn, move, zoom

video 0:38

1. Now look round the part.
2. Hold Shift and drag with the left button to turn the view.
3. Drag with the middle button, to move the view sideways.

   Roll the wheel forward to zoom.

4. Click a face of the view cube, to look along that axis.



The screen at 0:52 in the video.

4 Display buttons, Save is not Export

video 0:52

The display buttons, right of the ribbon, set how the model is drawn.

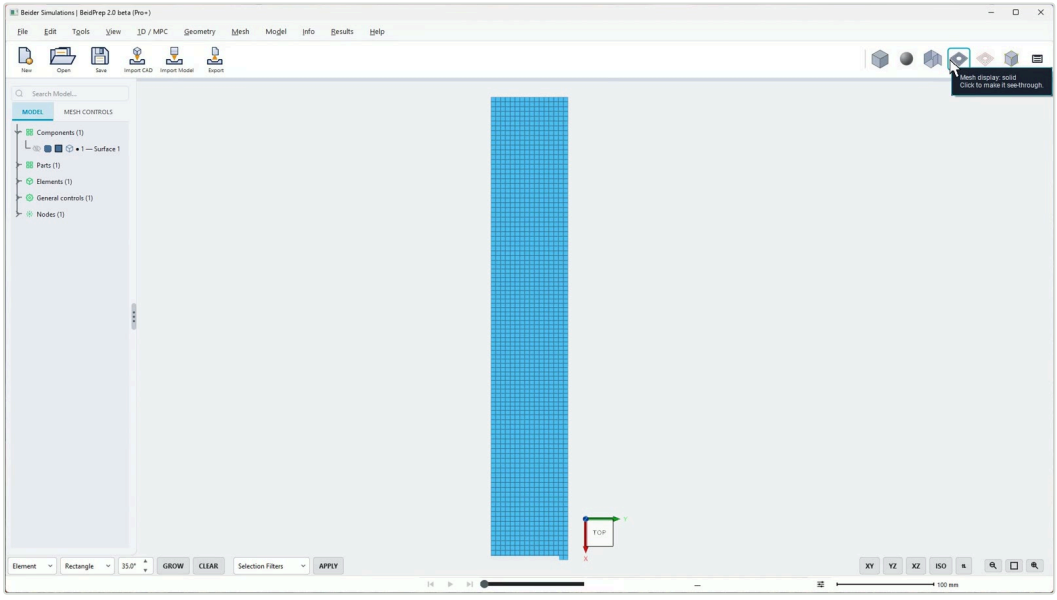
Each click moves to the next setting, and the tooltip says what the next click does.

Save is not Export.

Export writes the deck that OpenRadioss runs.

Save writes a BeidPrep model, a dot beid file, with the CAD, the mesh, its meshing settings and the cards.

This model came from a dot beid file, so Control S saves straight to it.



The screen at 0:56 in the video.

5 Open a deck

video 1:18

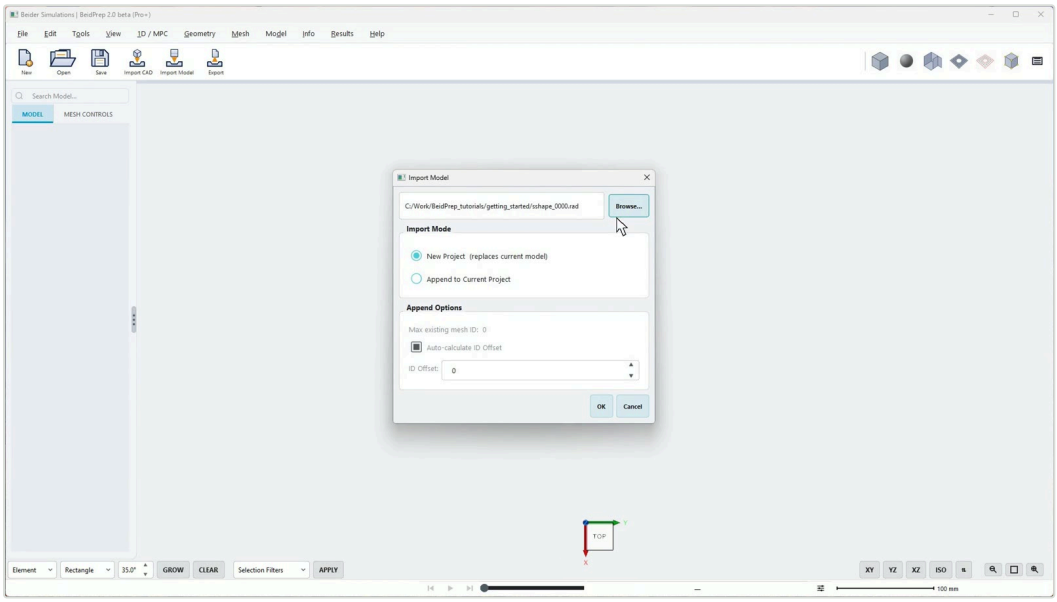
1. Now open the deck, as you would open any OpenRadioss model.
2. Click New on the File tab, and press Control I.
3. Click Browse, and choose the Starter file.
4. Leave New Project chosen, and click OK.

   If an Engine file sits in the same folder, it is opened with it.

   Append to Current Project would add the deck, to the model already open.

   Its numbers are raised by an offset, so they do not clash.

   In the video: **sshape_0000.rad** in file dialog: File name box.



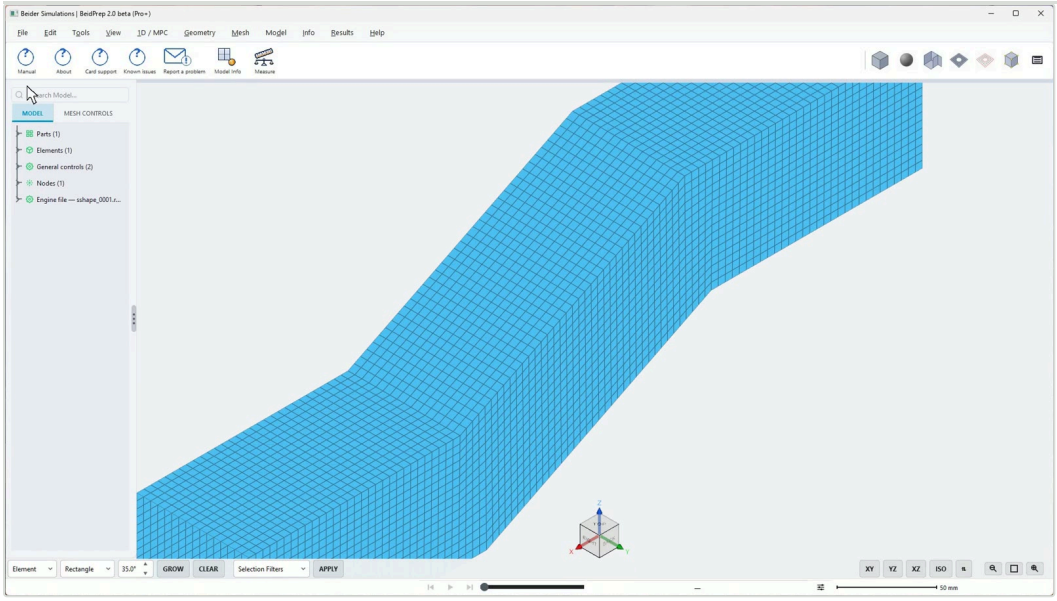
The screen at 1:30 in the video.

6 The manual, Close

video 1:45

1. Open the manual at any time with Help > Manual, the F1 key, or the Manual button on the ribbon.

   The next videos take each part in turn: the model tree, the card editor, and the way from CAD to a shell mesh.



The screen at 1:52 in the video.

Steps and pictures are taken from tutorial video 01. Every picture is a frame of the video.

TUTORIAL VIDEO 02

02 The model tree, visibility and groups

Find your way round the model tree: search it, hide and isolate parts, change a part's colour, and build a node group from a selection in the 3D view -- with undo for every step.

Tutorial files	folder `02_Model_tree_visibility_groups` — open `sshape_0000.rad` first
Video	Watch video 02 on YouTube
Download files	Download the tutorial files (97 MB)

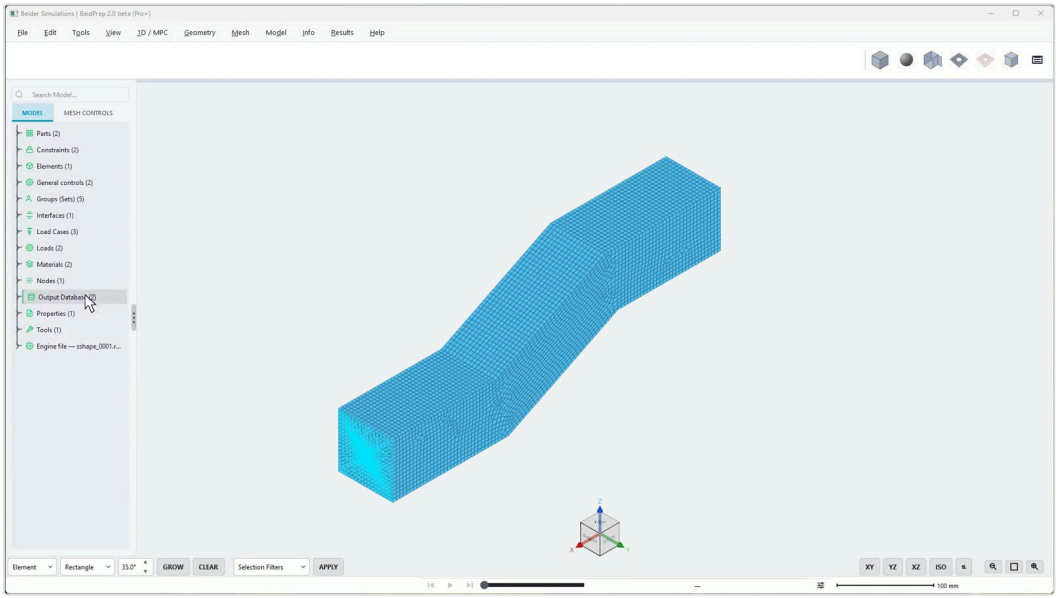
1 Opening, The tree

video 0:00

This video shows the model tree.

You will find cards, show and hide parts, and fill a group from the 3D view.

The model tree lists every card of the deck, sorted by category.



The screen at 0:11 in the video.

2 Search, Clear, Eye

video 0:11

1. Type in the search box above the tree.

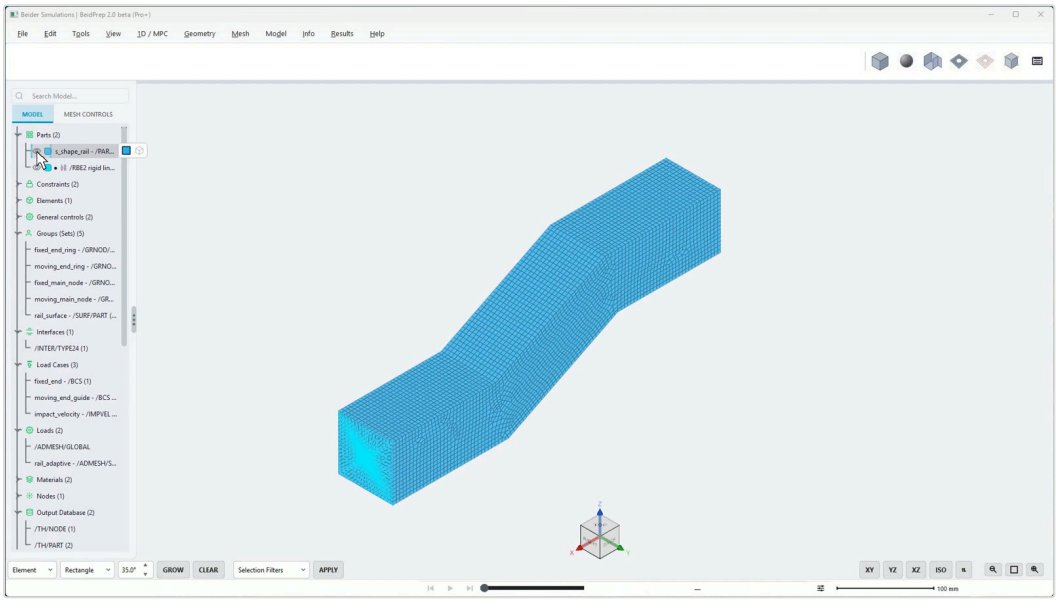
   Only the rows that match are kept.

   Clear the box, and the whole tree is back.

   Each part row carries an eye.

2. Click it to hide the part, and click again to show it.

In the video: **part** in Search box.



The screen at 0:24 in the video.

3 Isolate, Show All, Change Color

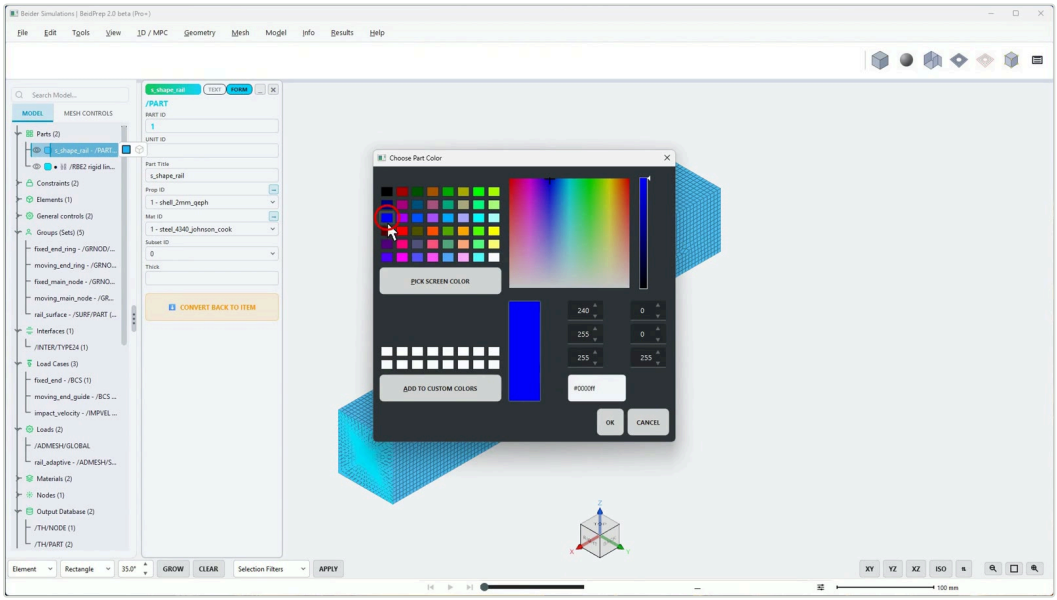
video 0:24

A right click on a part offers more.

Isolate shows this part only.

Show All brings every part back.

Change Color picks the colour the part is drawn in.



The screen at 0:35 in the video.

4 Hiding is only view, Card editor, Groups

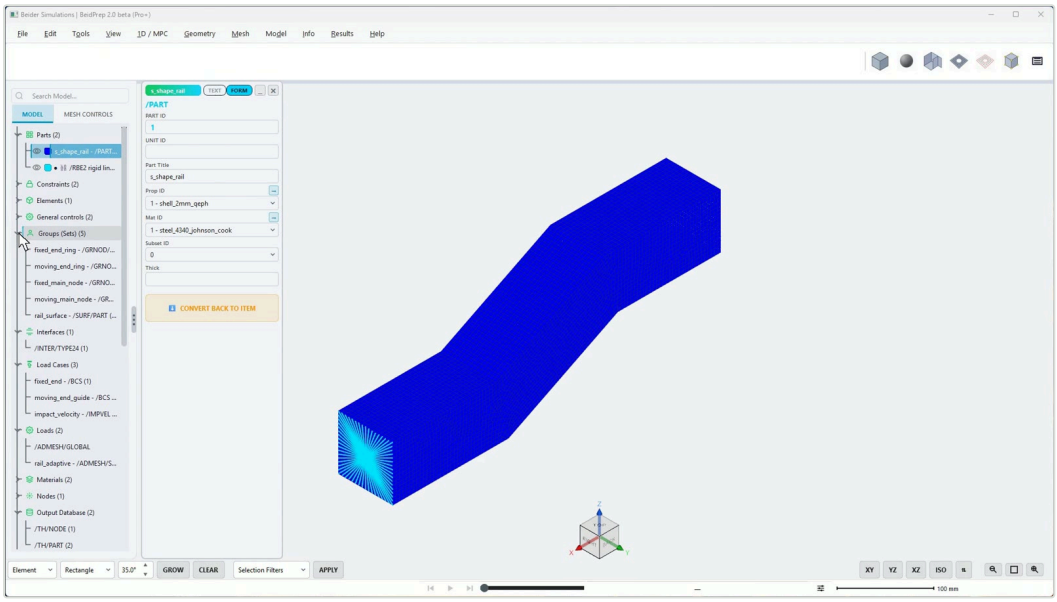
video 0:36

Hiding changes only what you see.

Hidden parts stay in the deck, and are exported as usual.

1. Click a card in the tree, and it opens in the card editor, beside the tree.

   Groups collect nodes, elements, parts or surfaces under one number, so a load or a contact can name them all at once.



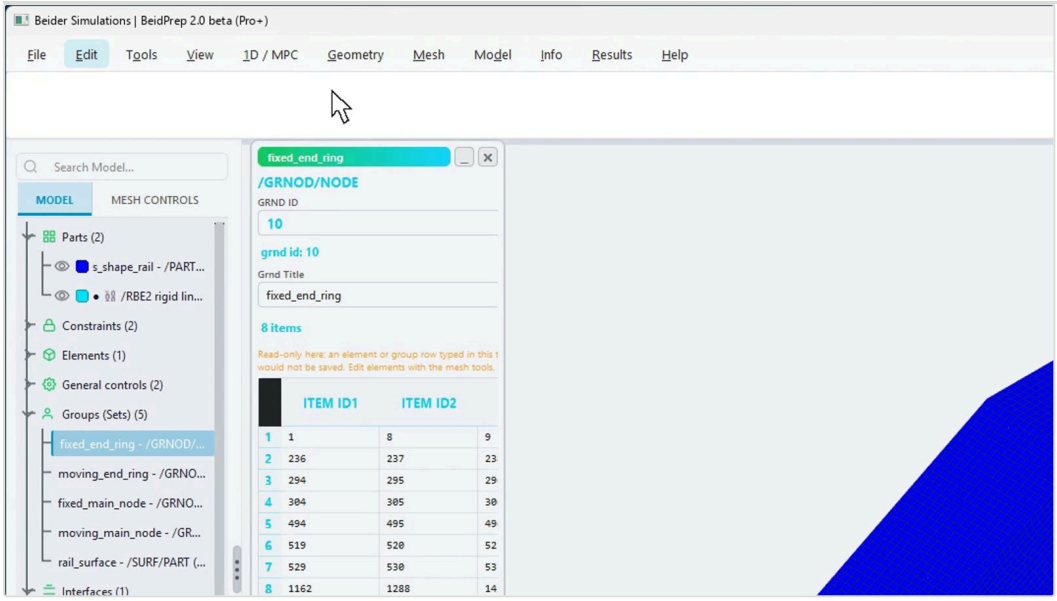
The screen at 0:54 in the video.

5 Node group, Select nodes, Add from Selection

video 0:54

1. Open the node group of the fixed end.
2. Set the selection bar to Node, and select nodes in the 3D view.
3. Click Add from Selection.

   Nine nodes go in, on a new row: the Edit menu offers to undo adding them.



The screen at 1:09 in the video.

6 Read-only table, Undo

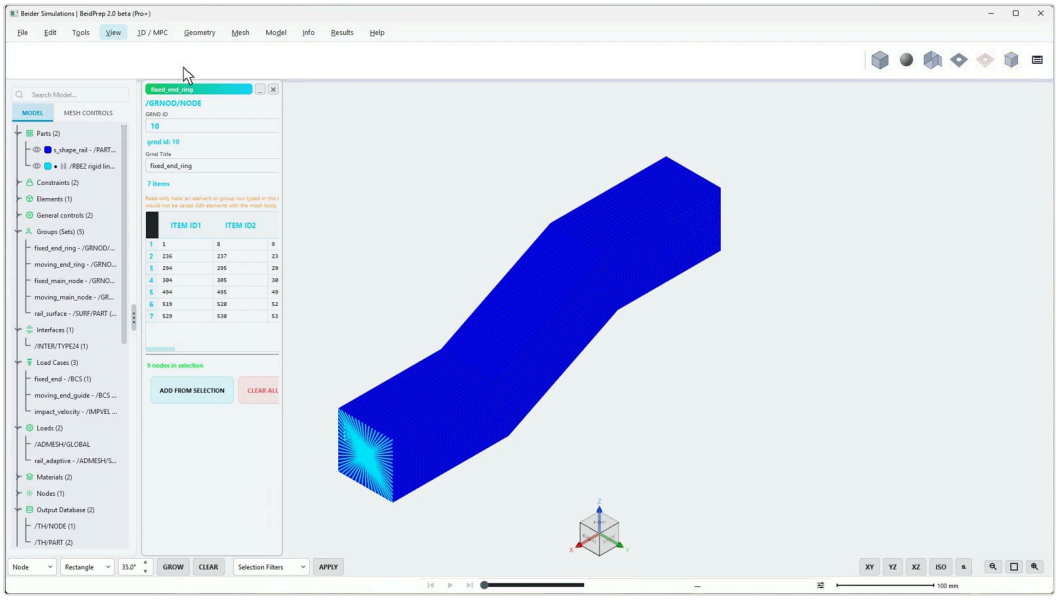
video 1:09

The table itself is read-only.

A number typed into it would not be saved, as the note above it says.

Nodes go into a group from the 3D view.

Control Z takes them out again.



The screen at 1:21 in the video.

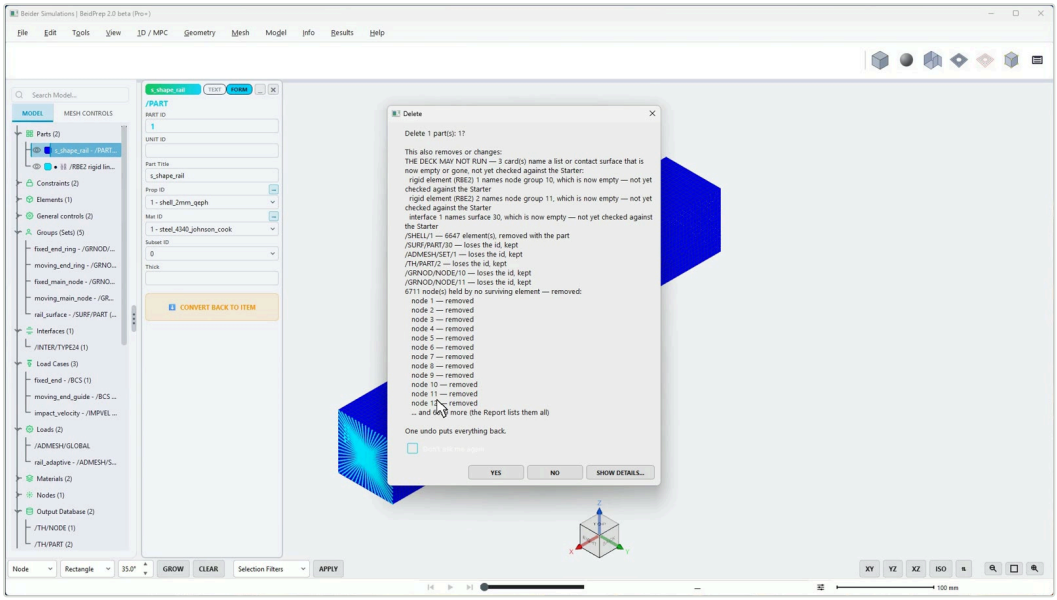
7 Delete asks first

video 1:22

1. Select a part in the tree and press Delete.

This delete would break the deck, so BeidPrep stops and asks.

The box lists what else goes, and warns that three cards would name an empty group or surface.



The screen at 1:34 in the video.

8 Delete with Yes, Undo the delete

video 1:34

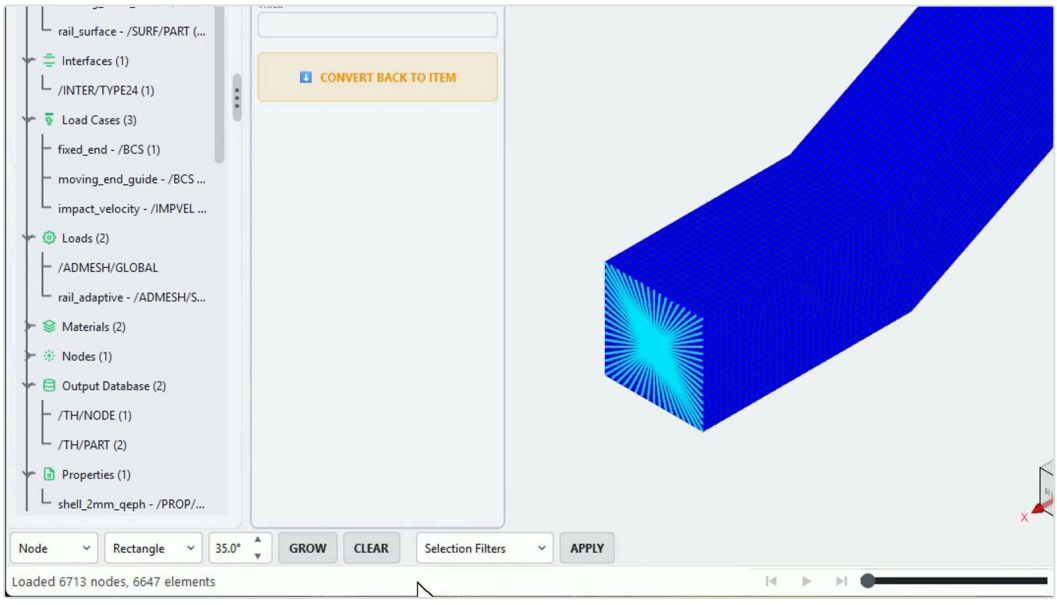
1. Leave Don't ask me again unticked, so the box keeps guarding the deck.

   Yes removes the part, with every node and element that only it used.

   The status line says what went.

   Control Z brings everything back, as the box said.

   Each of these edits is one step on the undo list.



The screen at 1:53 in the video.

Steps and pictures are taken from tutorial video 02. Every picture is a frame of the video.

TUTORIAL VIDEO 03

03 The card editor

Edit any card in the card editor: typed fields, drop-down lists that explain each value, references between cards, the function editor with its non-linearity check, and undo.

Tutorial files	folder `03_Card_editor` — open `sshape_0000.rad` first
Video	Watch video 03 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, Open a card

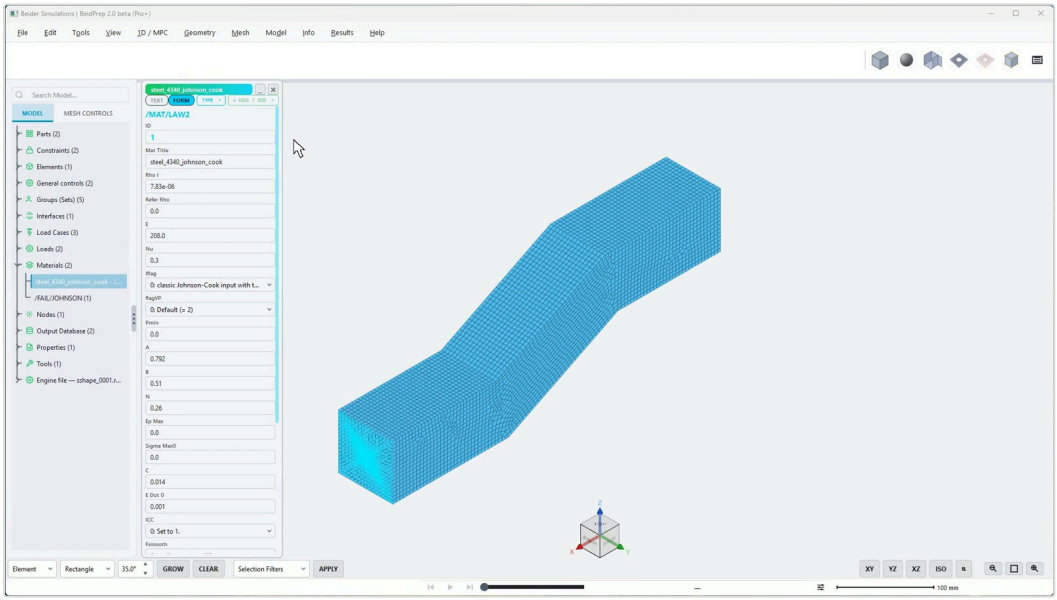
video 0:00

This video shows the card editor.

You will open a card, change it safely, and undo it.

1. Click a card in the model tree.

   The card editor shows its keyword, its title, and its fields, in the rows the solver reads.



The screen at 0:15 in the video.

2 Descriptions, Type a value, Drop-down

video 0:15

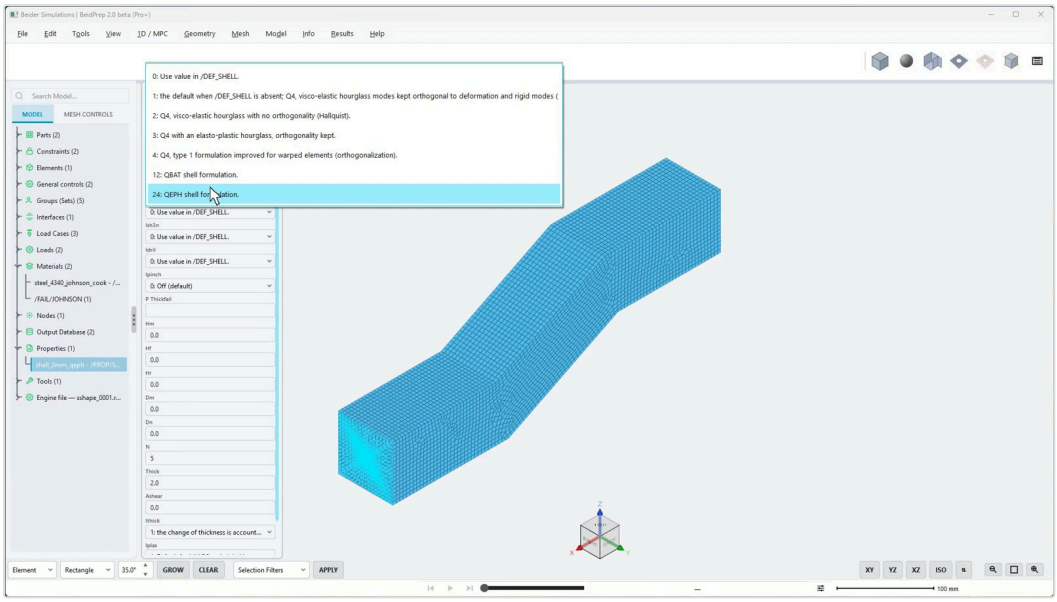
Each field has a short description of what it means.

1. Type a value and press Enter.

   It is written into the card in the columns the solver reads it from.

   A field with a fixed set of values is a drop down list, and each entry says what it means.

In the video: **0.8** in yield stress field.



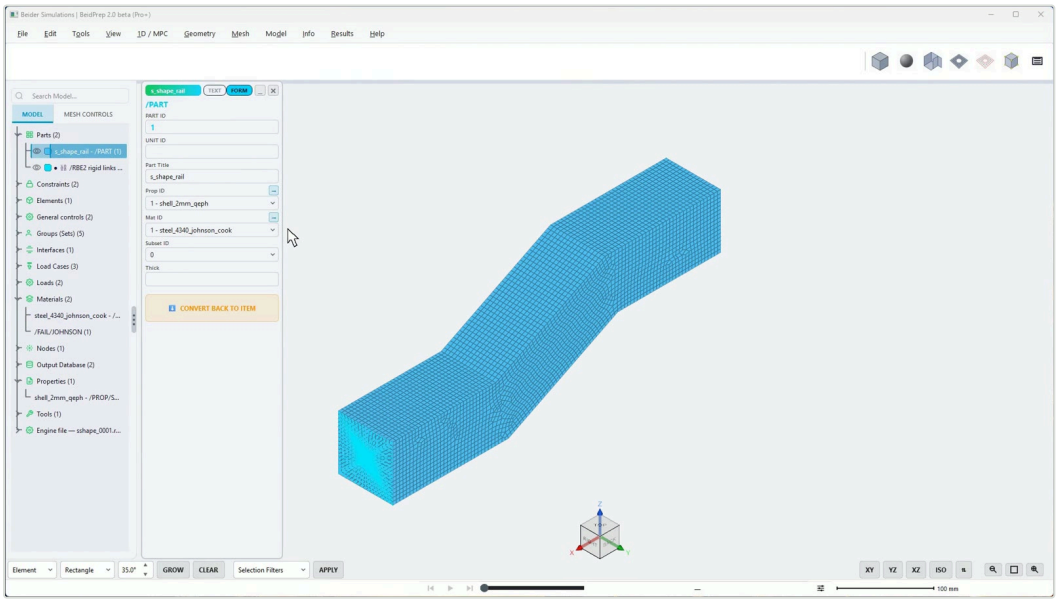
The screen at 0:31 in the video.

3 Options change rows, References

video 0:31

Some options change the card itself: the rows that belong to them appear and disappear.

A field that names another card, a material or a property, takes that card's number.



The screen at 0:41 in the video.

4 Own number, Function Editor, Table and plot

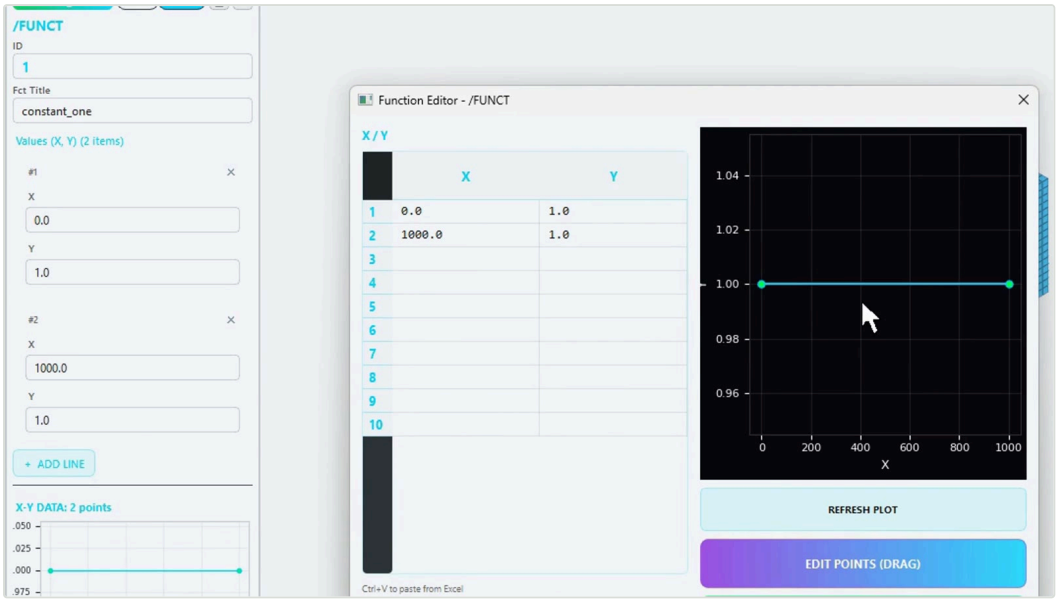
video 0:41

The card's own number cannot be typed over.

BeidPrep keeps every reference to it in step.

A curve card opens the Function Editor.

The points sit in a table beside their plot.



The screen at 0:53 in the video.

5 Non-linearity check

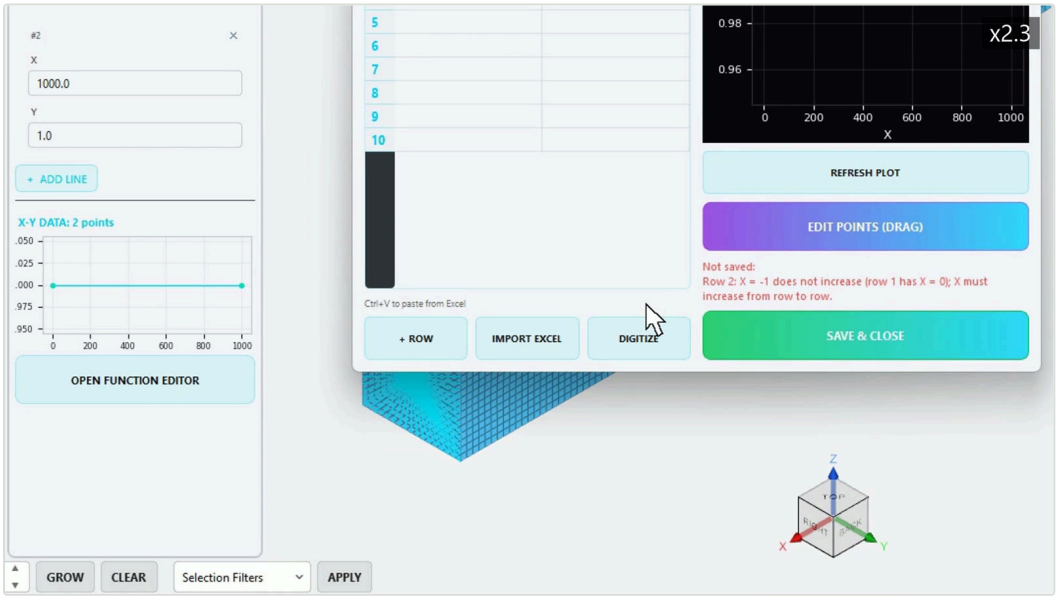
video 0:53

Save and Close writes the points only when every X value is larger than the one before.

Y may rise and fall freely, like a sine-wave acceleration.

Otherwise it says Not saved, and names the row.

In the video: -1 in x cell.



The screen at 1:06 in the video.

6 Put it back, Undo

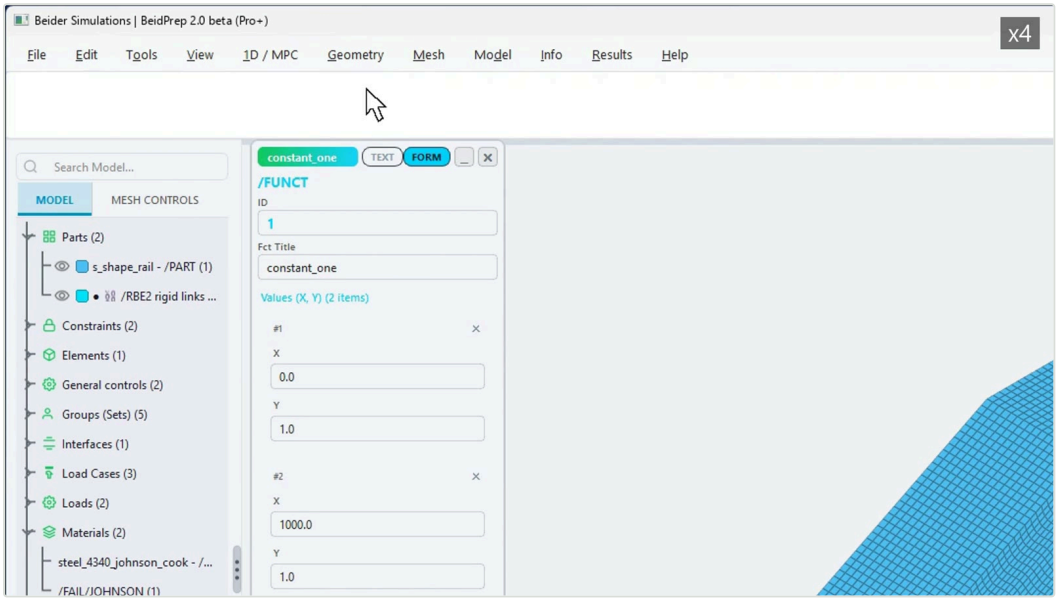
video 1:06

1. Put the value back, and the curve saves.

   Every edit can be undone with Control Z.

   The Edit menu names the edit it will undo.

   In the video: **1000** in x cell.



The screen at 1:16 in the video.

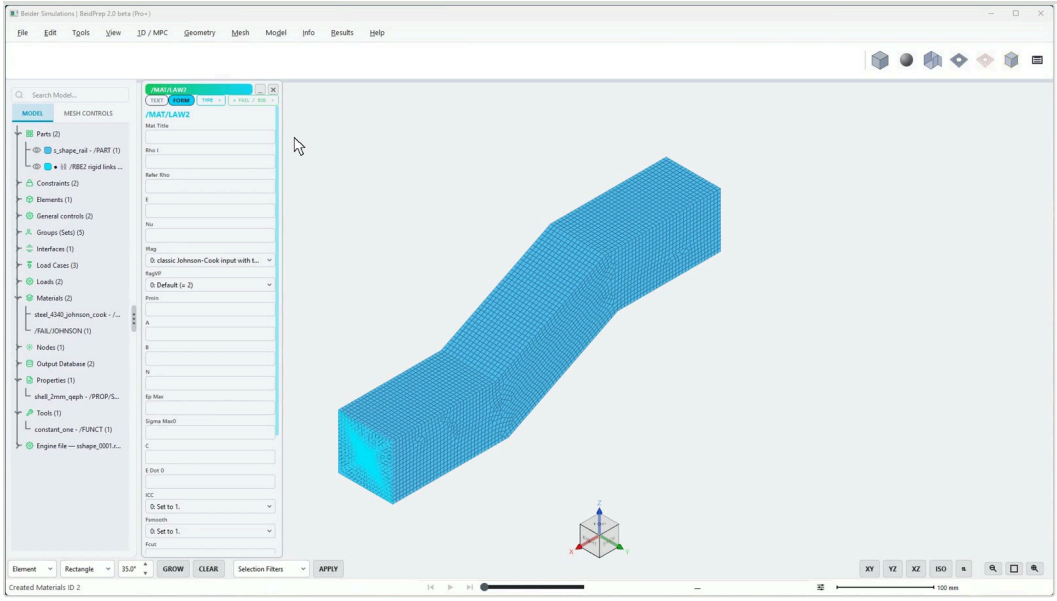
7 Create a card, Undo it

video 1:16

To create a card, right click an empty place in the tree, and choose its type.

It opens with the next free number.

1. Undo it, and the deck is as it was.



The screen at 1:26 in the video.

Steps and pictures are taken from tutorial video 03. Every picture is a frame of the video.

TUTORIAL VIDEO 04

04 From CAD to shell mesh: midsurface, thickness parts, mesh check

Take a CAD part to a shell mesh: import a STEP file, check how its faces connect, extract the mid-surface -- each wall thickness becomes its own part at once -- clean it up, mesh it, and read the mesh check.

Tutorial files	folder `04_CAD_to_shell_mesh` — open `part2_carrier_tray.step` first
Video	Watch video 04 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, Import CAD

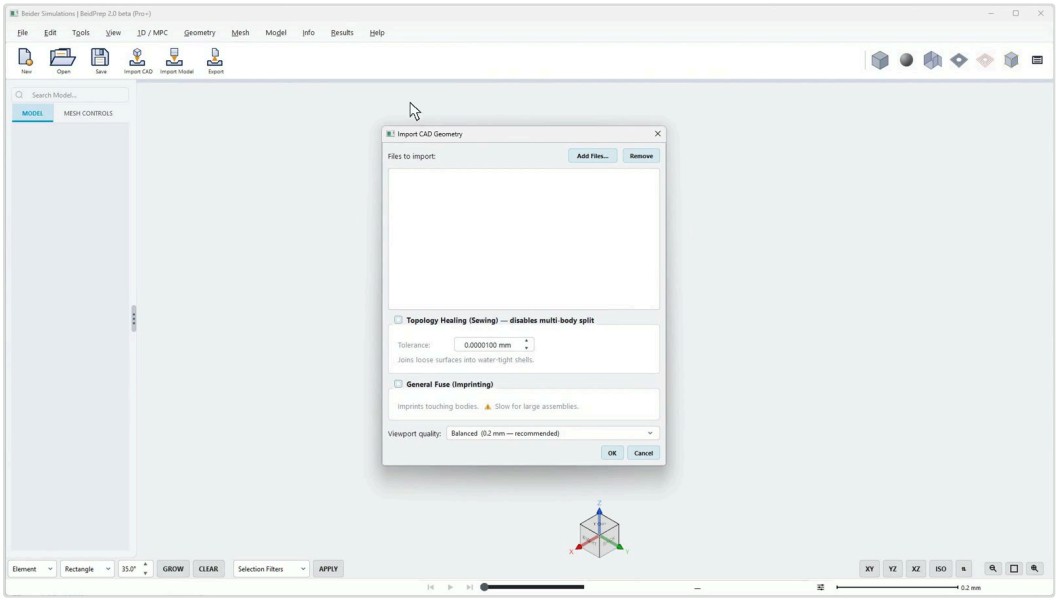
video 0:00

This video takes a CAD part all the way to a shell mesh.

You will import a STEP file, make its mid-surface, mesh it, and check the mesh.

1. Click Import CAD on the File tab.

   The Import CAD Geometry window opens.



The screen at 0:14 in the video.

2 Add files, Options

video 0:15

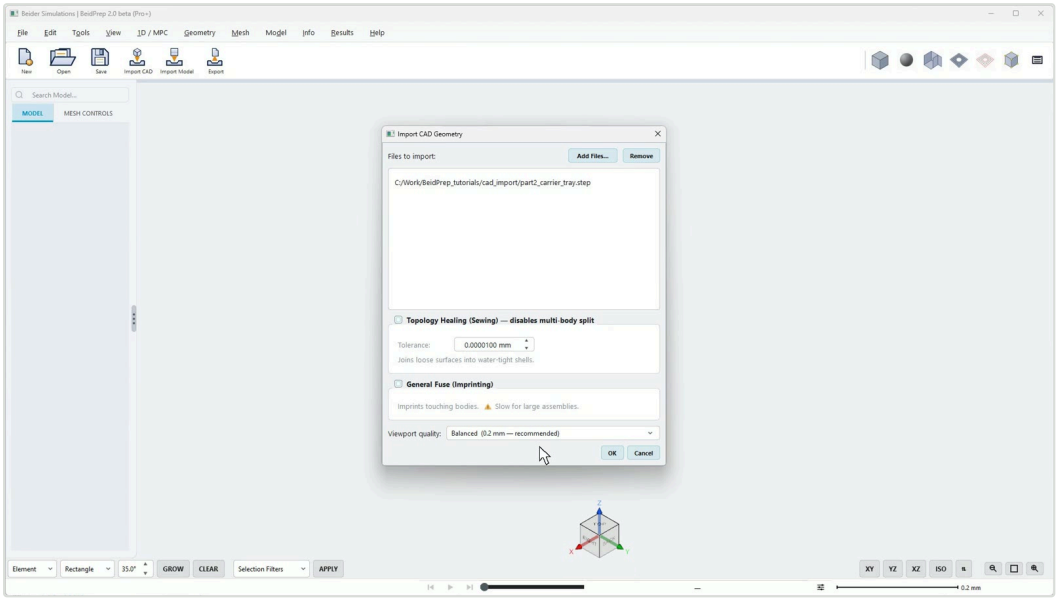
1. Click Add Files, and choose the STEP file.

   You can add several files; they are read one after the other.

Sewing and imprinting stay off for a single clean solid like this one.

Viewport quality changes only how the part is drawn, never the geometry you mesh.

Balanced is the recommended setting.



The screen at 0:34 in the video.

3 Import, Look round

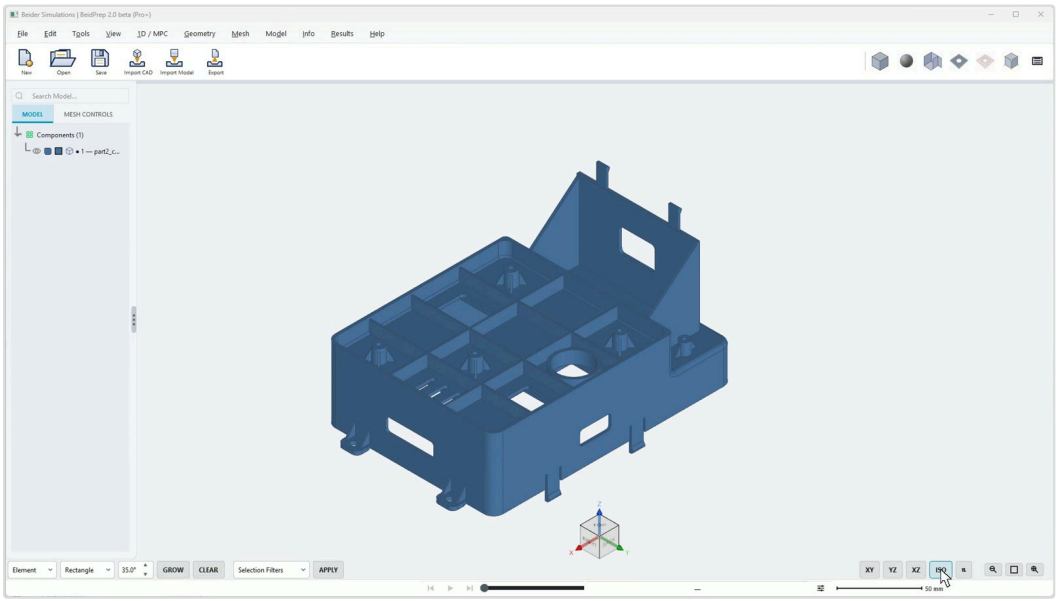
video 0:34

1. Click OK.

   The tray appears in the 3D view, and as a CAD body in the model tree.

2. Turn the part with the left mouse button, and zoom with the wheel.

   It is a thin-walled tray, with ribs, bosses, cut-outs and mounting feet.



The screen at 0:50 in the video.

4 Edges by connection

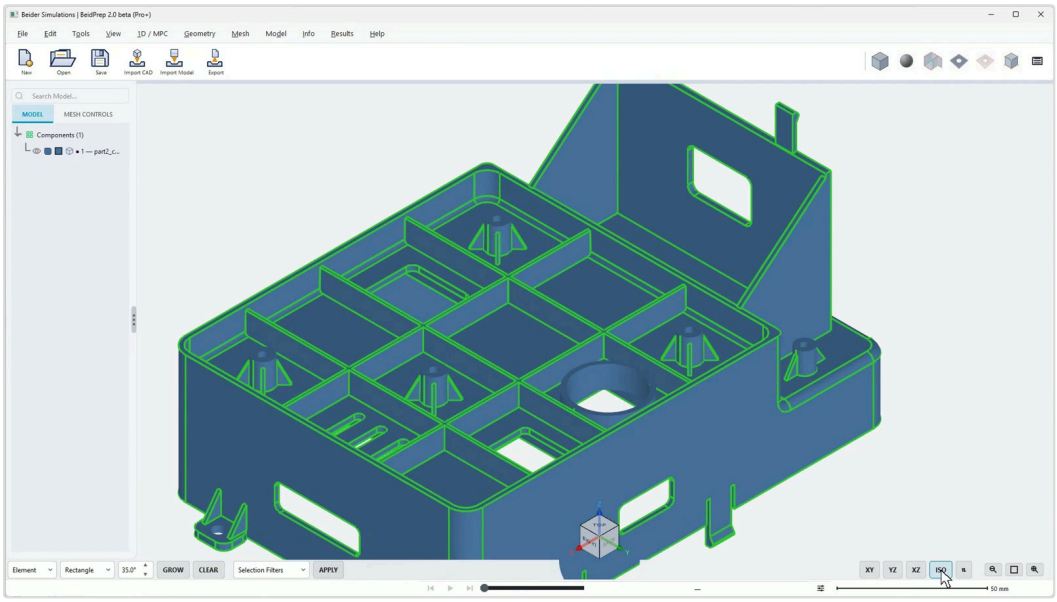
video 0:50

The display buttons change how CAD is drawn.

This one colours the edges by how the faces connect.

Green is an edge shared by two faces. Red is an open edge: a gap you would have to close before meshing.

This part is all green.



The screen at 1:06 in the video.

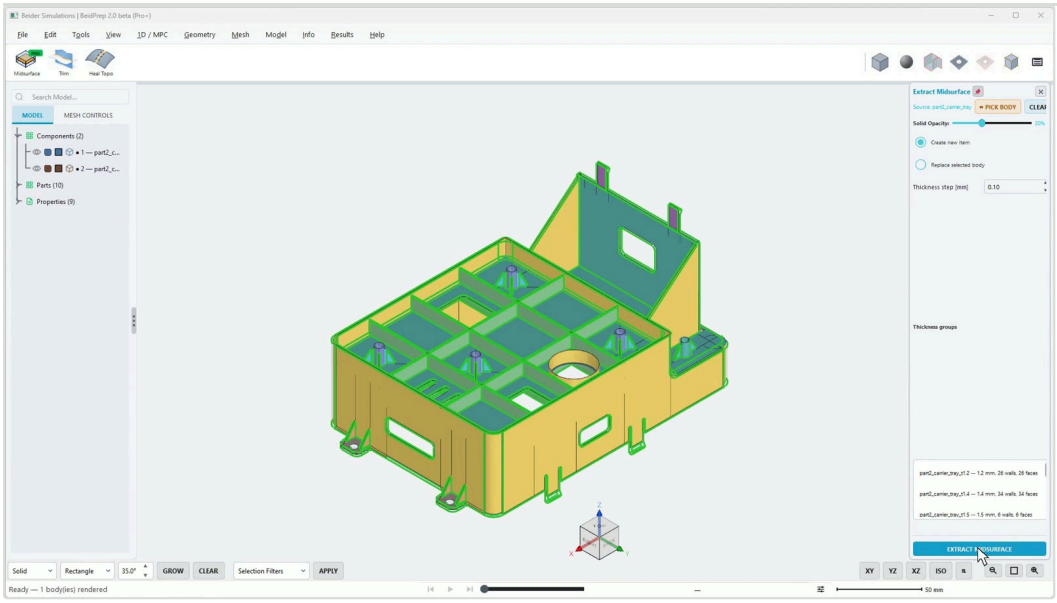
5 Midsurface

video 1:07

The tray is a solid, but its walls are thin.

A thin wall is meshed with shells on its mid-surface.

1. Click Midsurface on the Geometry tab, and click the solid.
2. Keep Create new Item, and click Extract Midsurface.



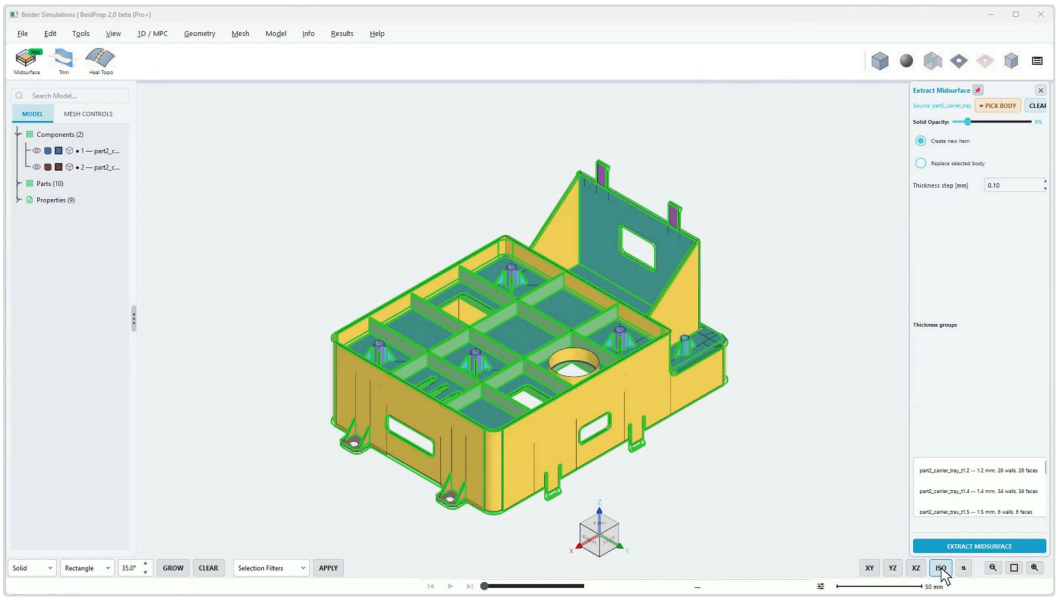
The screen at 1:20 in the video.

6 What it built

video 1:21

Each wall gets its own mid-surface, halfway between its two faces.

Where walls meet, their mid-surfaces share an edge, so the mesh on them will be connected.



The screen at 1:32 in the video.

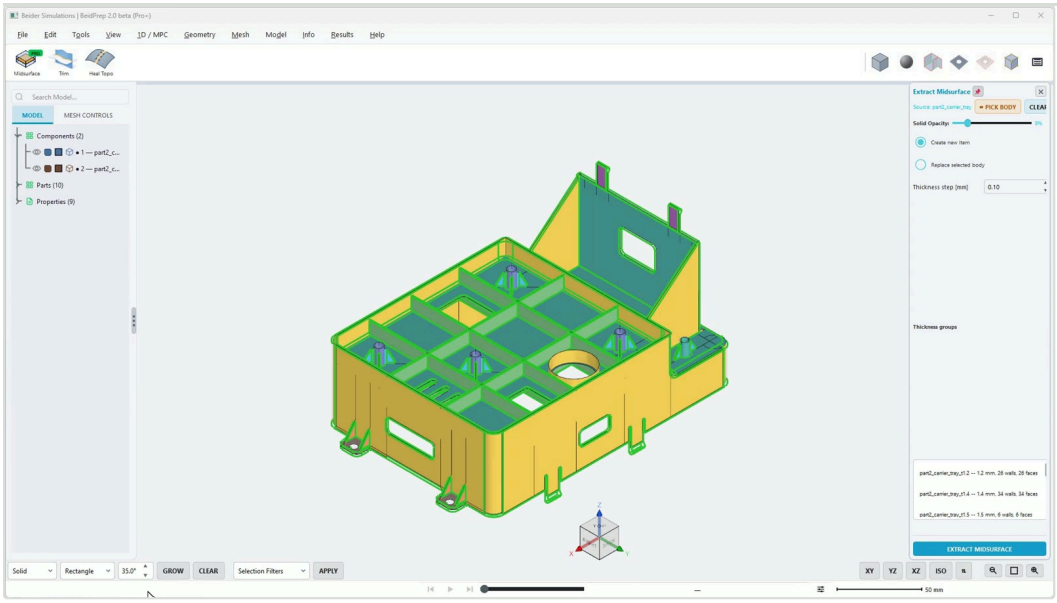
7 Nothing left out

video 1:32

The tool does not guess.

A region it cannot read as a wall is left out, drawn orange, and listed here with its reason.

On this part, nothing is left out.



The screen at 1:43 in the video.

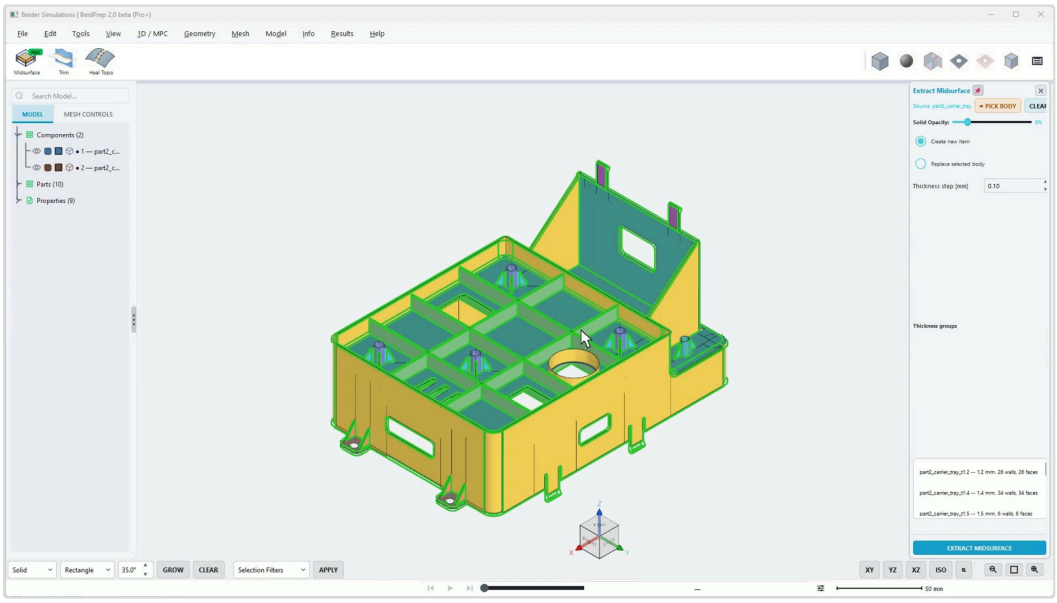
8 Thickness groups

video 1:43

Thickness groups lists the walls by thickness.

Each wall's thickness is measured, and rounded to the Thickness step.

Each group is already its own part, drawn in its own colour.



The screen at 1:54 in the video.

9 Parts in the tree, Heal Topo

video 1:54

1. Widen the model tree, and open Parts.

   Each part is named after the tray and its thickness, and its colour matches the 3D view.

Before meshing, Heal Topo on the Geometry tab has clean-up tools.

Consolidate Fragments removes small loose faces left round holes, and keeps every hole at its exact size.

2. Choose the mid-surface body.

   A note says what was removed, and that every hole was kept.



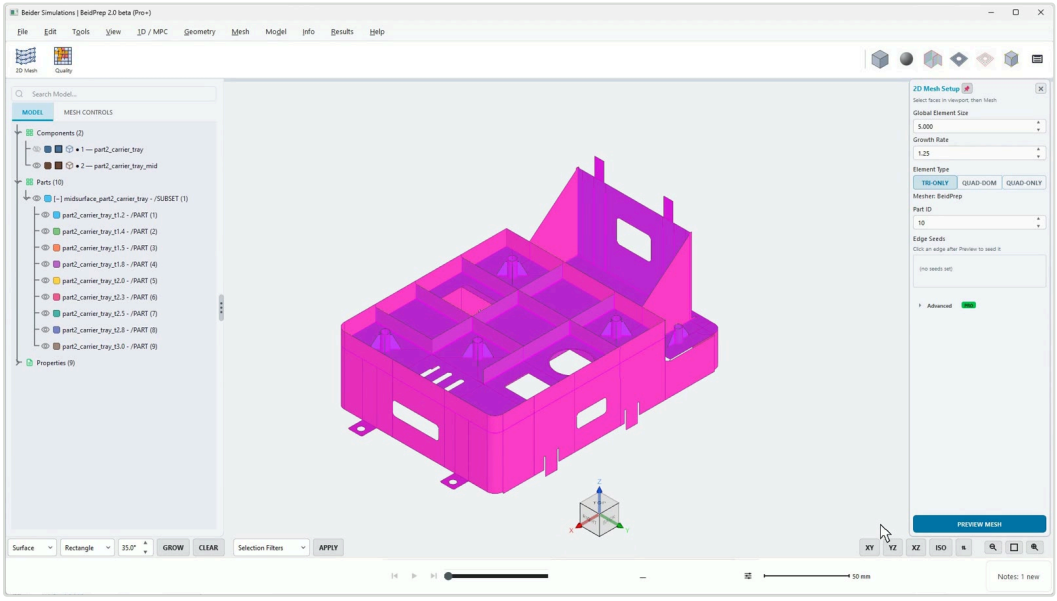
The screen at 2:22 in the video.

10 Hide the solid, 2D Mesh

video 2:22

1. Hide the solid in the model tree, so only the mid-surface is shown.
2. Now click 2D Mesh on the Mesh tab, and drag a rectangle around the whole mid-surface.

   Every face turns magenta.



The screen at 2:33 in the video.

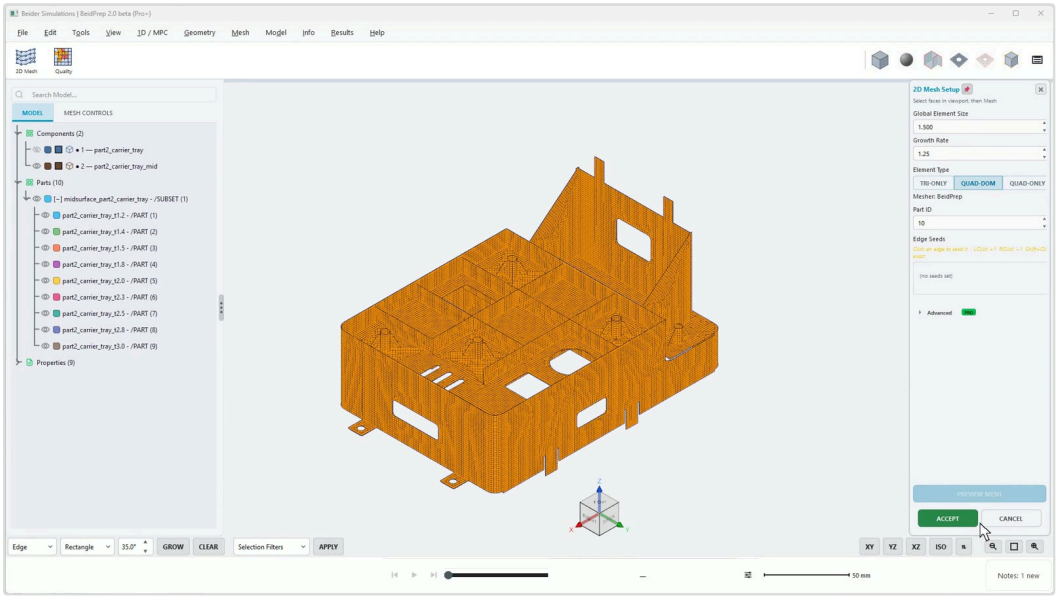
11 Size and type, Preview

video 2:33

1. Set the element size for this part.
2. Choose Quad Dom.
3. Click Preview Mesh.

   The mesh is drawn in orange; it is not in the model yet.

   In the video: **1.5** in Global Element Size.



The screen at 2:46 in the video.

12 Accept, Mesh check

video 2:47

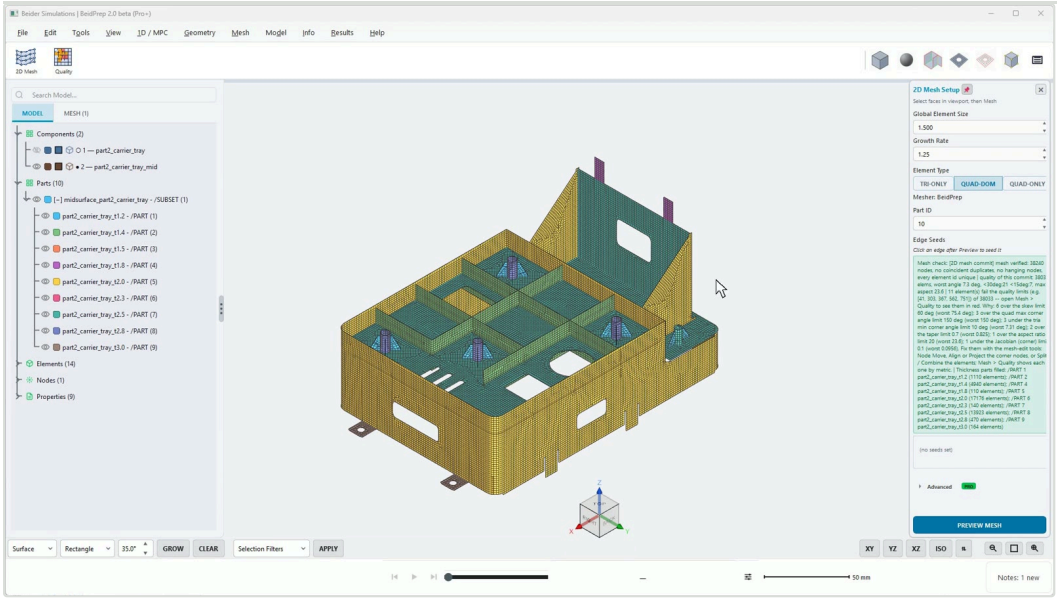
1. Click Accept.

   The mesh goes into the model, and each wall's elements go into its thickness part.

   Every accepted mesh is checked at once.

   It confirms that the mesh is connected: no duplicate nodes, and no hanging nodes.

   It also names the few elements that fail the quality limits; video six shows how to fix elements like these.



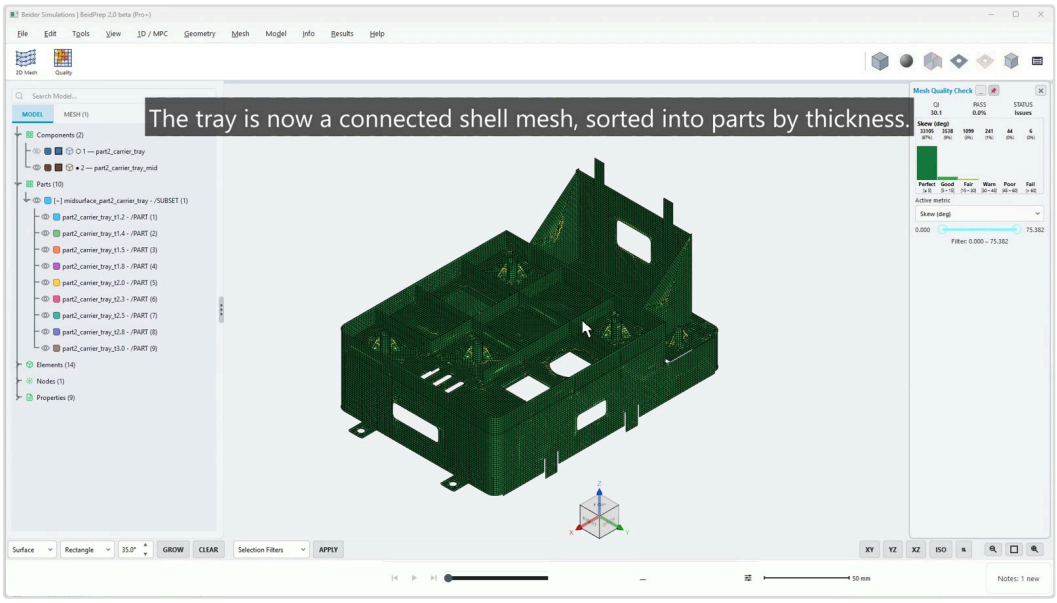
The screen at 3:08 in the video.

13 Quality, Close

video 3:08

1. Click Quality on the Mesh tab for the details.
2. Choose a criterion, and the elements are coloured by it.

   The tray is now a connected shell mesh, sorted into parts by thickness.



The screen at 3:15 in the video.

Steps and pictures are taken from tutorial video 04. Every picture is a frame of the video.

TUTORIAL VIDEO 05

05 Mesh editing: nodes (move, morph, create, equivalence)

Edit a mesh node by node: move with or without morphing, merge on release, create nodes, and close a gap with node equivalence.

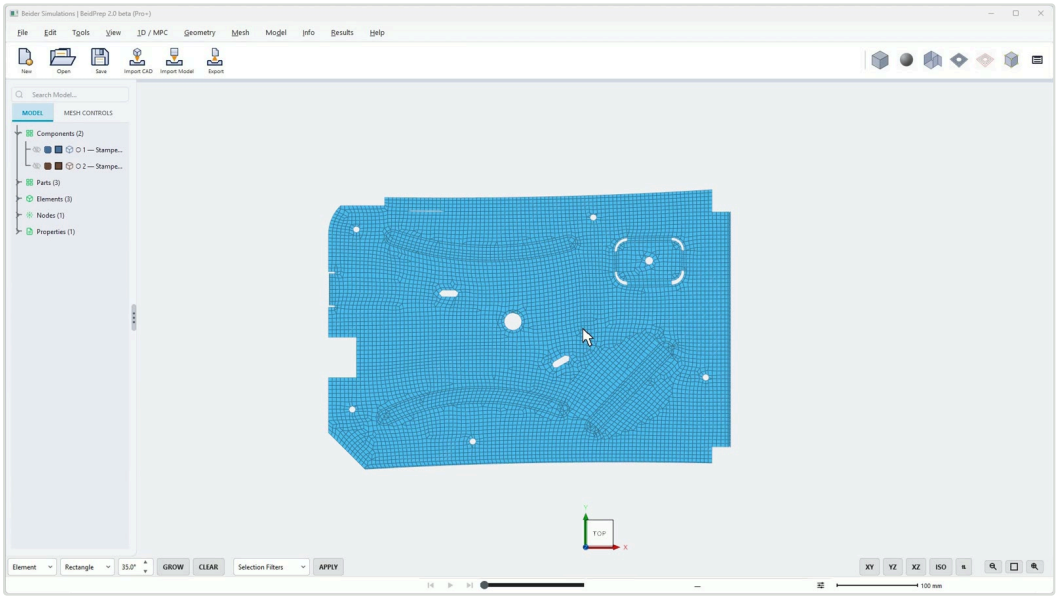
Tutorial files	folder `05_Mesh_editing_nodes` — open `sheet_gap.beid` first
Video	Watch video 05 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening

video 0:00

This video edits nodes on a real mesh: a stamped sheet-metal bracket, meshed from its midsurface.

The first mesh is good almost everywhere, but a few elements fail the quality limits, and a seam is not connected.



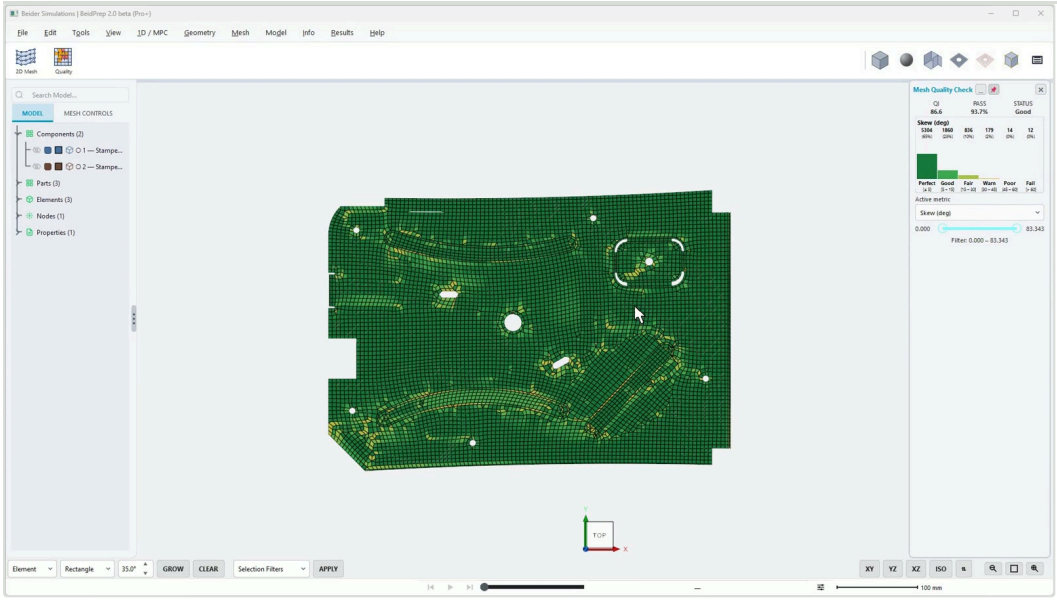
The screen at 0:13 in the video.

2 Quality colours

video 0:13

On the Mesh tab, Quality colours every element by a quality measure; here, skew.

Green is good. Yellow and red elements are the ones to fix.



The screen at 0:23 in the video.

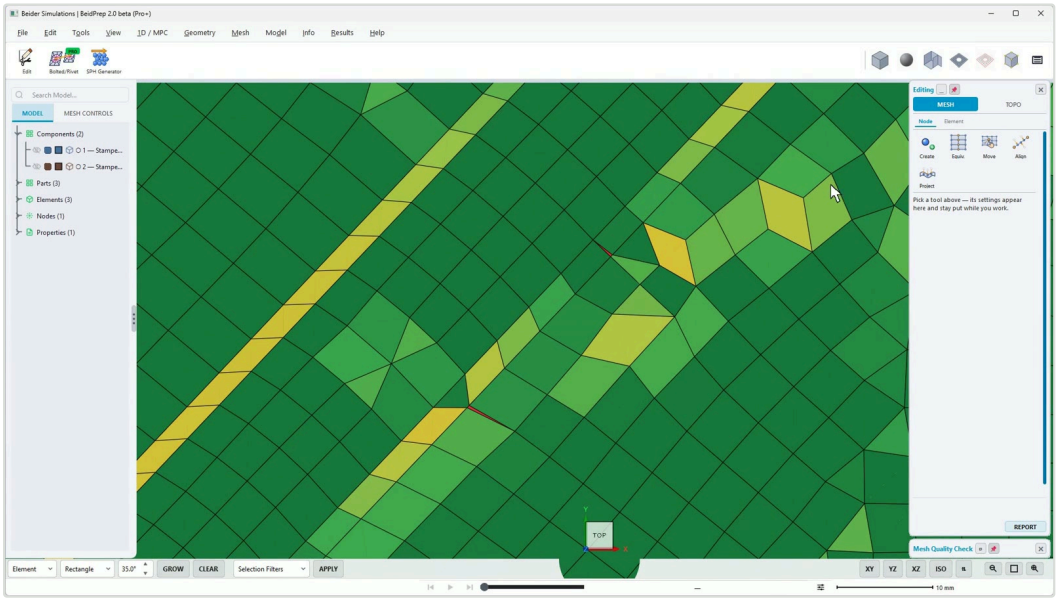
3 Editing toolbar, Area B

video 0:24

1. Open the Model menu, and click Edit.

   The node tools are Create, Equivalence, Move, Align and Project.

   Here, on the rib wall, is the worst element of the mesh: a sliver triangle with a corner of only three degrees.



The screen at 0:40 in the video.

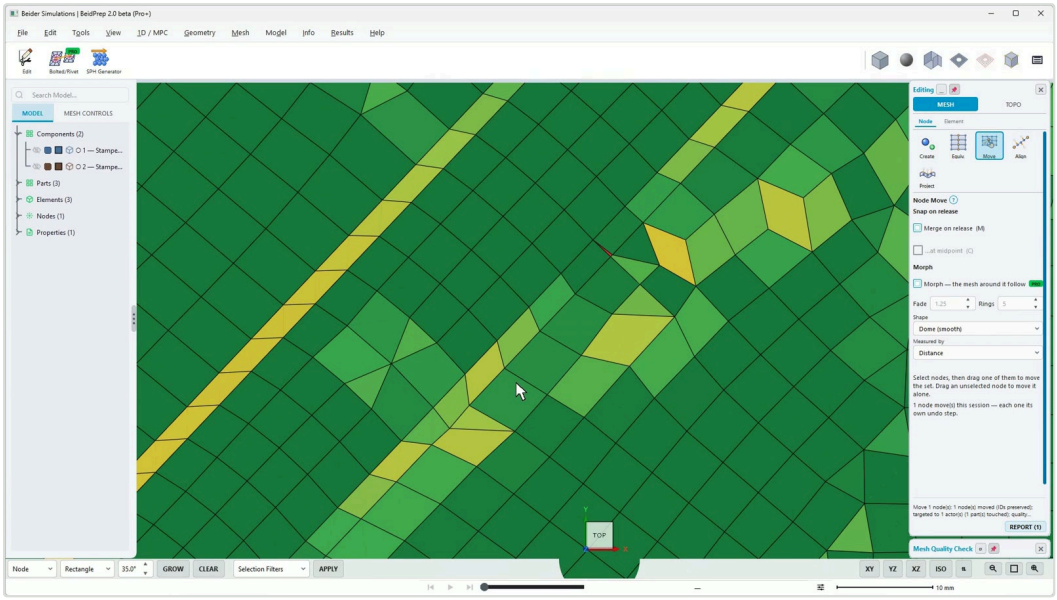
4 Move

video 0:40

1. Click Move, and drag one of its nodes along the surface.

The node slides in the plane of its elements, so it stays on the sheet.

The triangle opens up: its smallest corner goes from three degrees to about twenty eight, and it leaves the red.



The screen at 0:56 in the video.

5 Merge on release, Morph

video 0:56

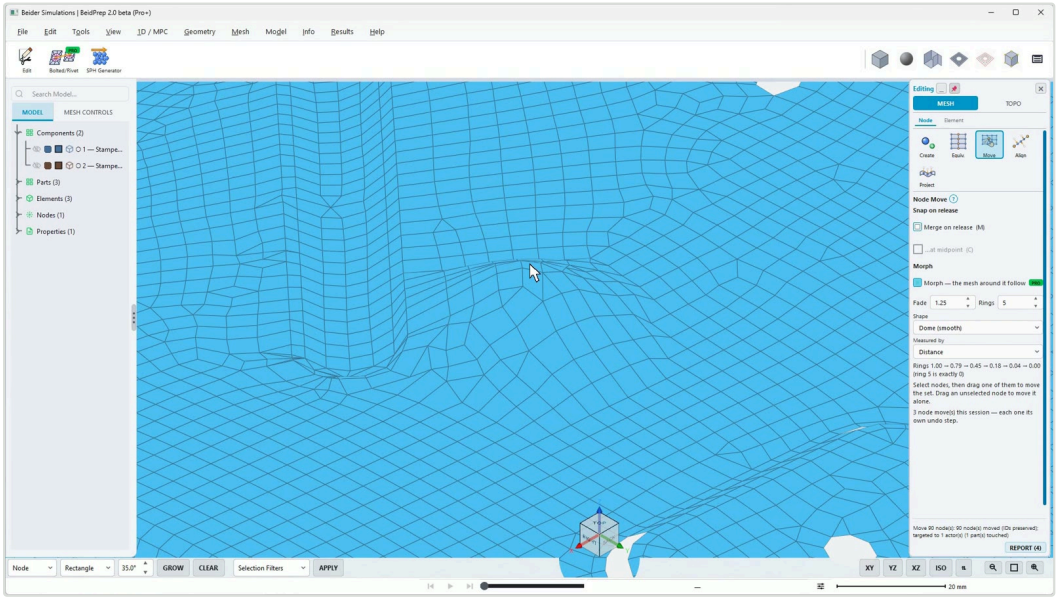
With Merge on release ticked, a node dropped on another node merges into it.

Control Z takes it back.

Morph moves the nodes around the one you drag as well, fading out over a number of rings.

1. Hold N and drag up, out of the sheet: the nodes around it follow, ring by ring, into a dome.

Control Z takes it back.



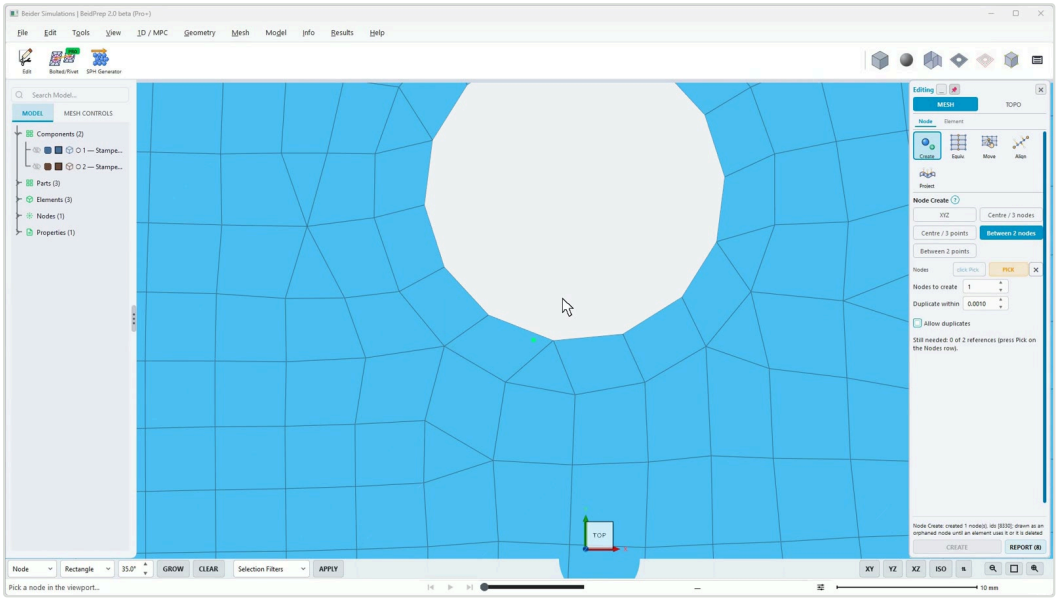
The screen at 1:22 in the video.

6 Create

video 1:25

Create adds a node: here, exactly halfway between two nodes you pick.

Control Z removes it again.



The screen at 1:39 in the video.

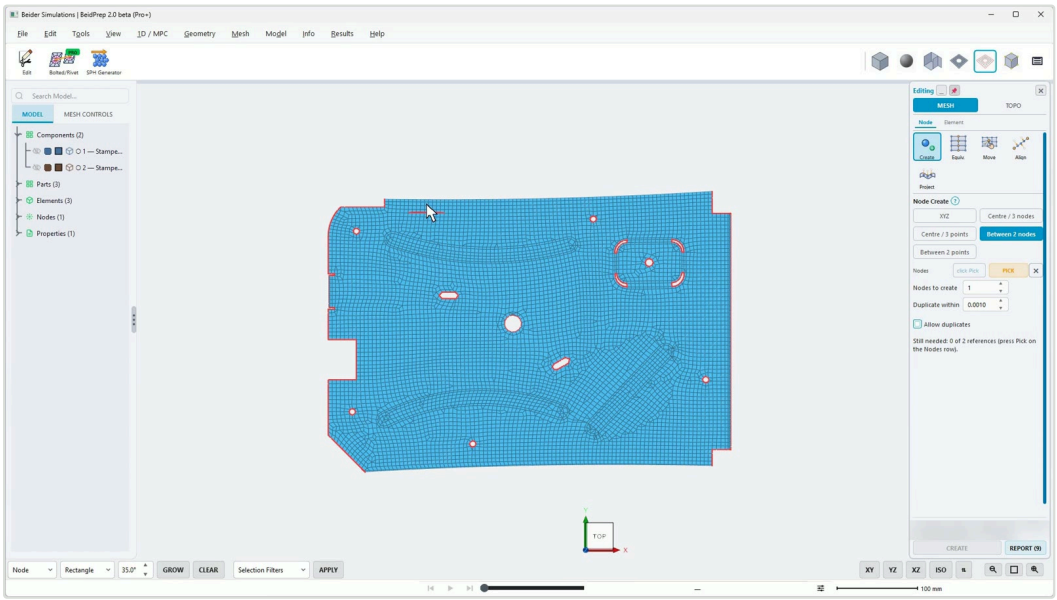
7 The seam

video 1:42

Now the seam. The Mesh Edges view draws, in red, every element edge used by only one element.

On a connected sheet that is only its outline and its holes.

This red line runs across the sheet: the elements on its two sides are not joined.



The screen at 1:58 in the video.

8 Zoom on the gap, Equivalence

video 1:58

Seen close up, the two rows of nodes are almost a millimetre apart.

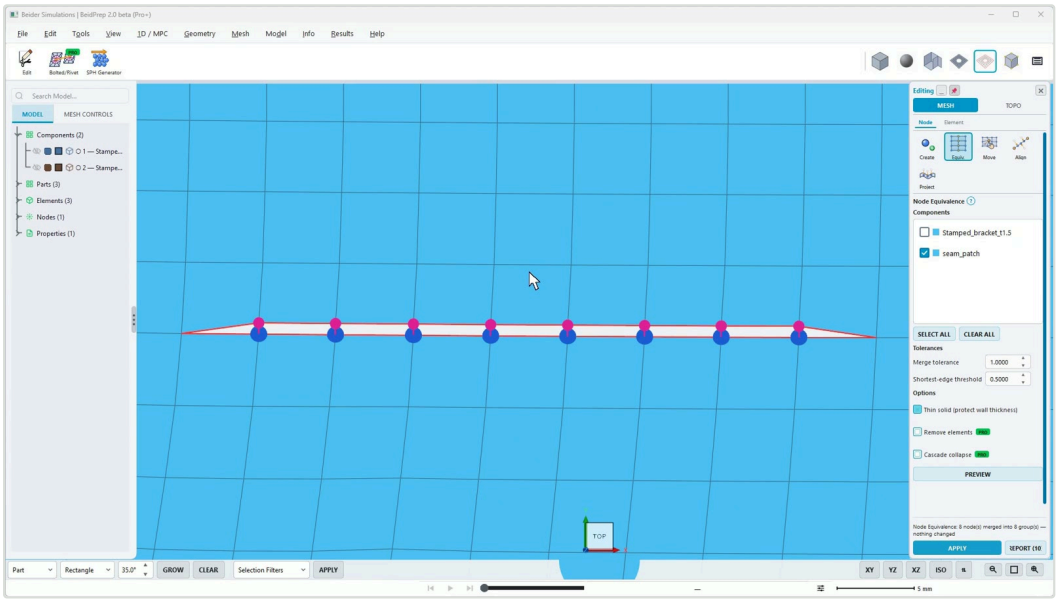
The solver would treat the two sides as separate pieces.

1. Click Equiv., tick only the part beside the seam, and set the merge tolerance to one millimetre.

Elsewhere on this sheet some nodes are meant to be closer than that, so the merge is kept to where it belongs.

Preview marks every pair of nodes that would merge, before anything changes.

In the video: **1** in Merge tolerance.



The screen at 2:23 in the video.

9 Apply, Align and Project, Close

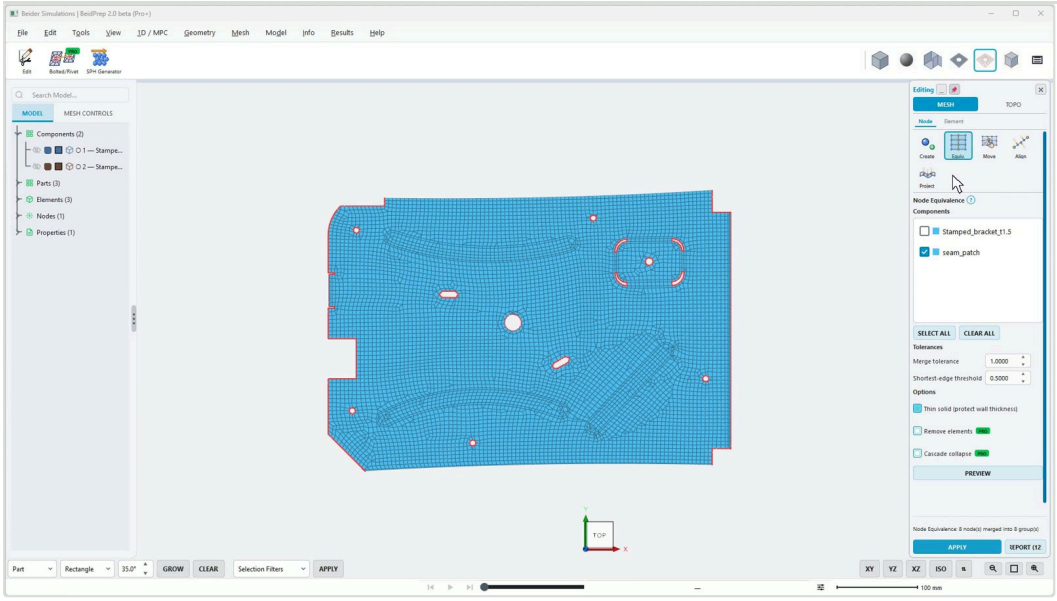
video 2:24

Apply merges them: the two rows become one.

The red edges are gone. The sheet is connected again.

Align lines nodes up between two end nodes, and Project moves nodes onto a surface, a plane or a line.

Every node edit is one step on the undo list, so you can try, look, and take it back.



The screen at 2:37 in the video.

Steps and pictures are taken from tutorial video 05. Every picture is a frame of the video.

TUTORIAL VIDEO 06

06 Mesh editing: elements (combine, split, fill, normals)

Edit a mesh element by element: combine two triangles into a better quad, split, delete and fill, move one node to clear a sliver, and make the shell normals of a whole area face the same way.

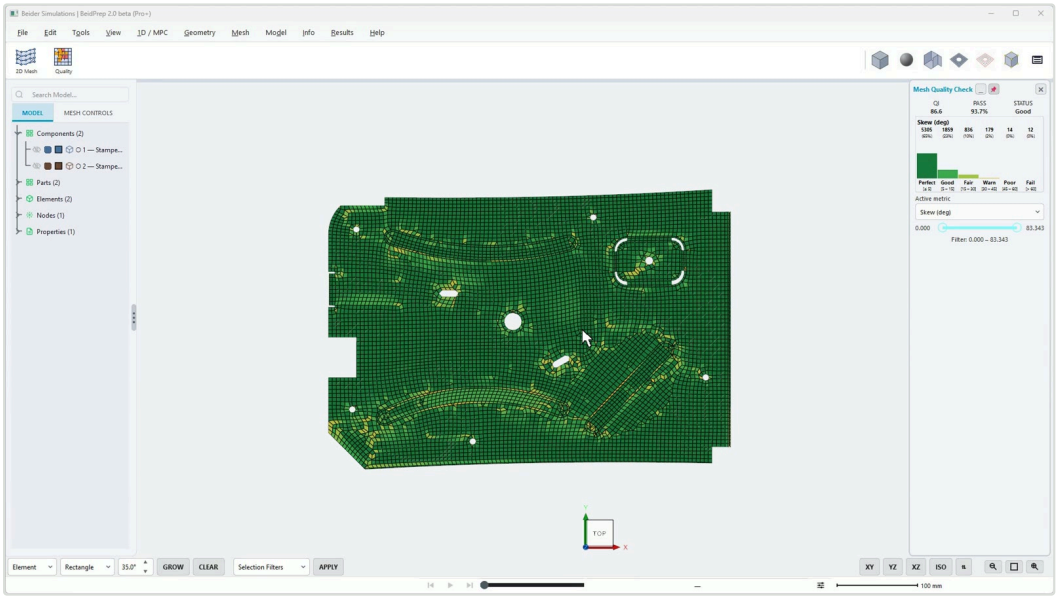
Tutorial files	folder `06_Mesh_editing_elements` — open `sheet_08.beid` first
Video	Watch video 06 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening

video 0:00

This video fixes poor elements on the same sheet-metal bracket, with the element tools.

You will see what combine and split can and cannot do, then delete and fill a poor patch, and check the result.



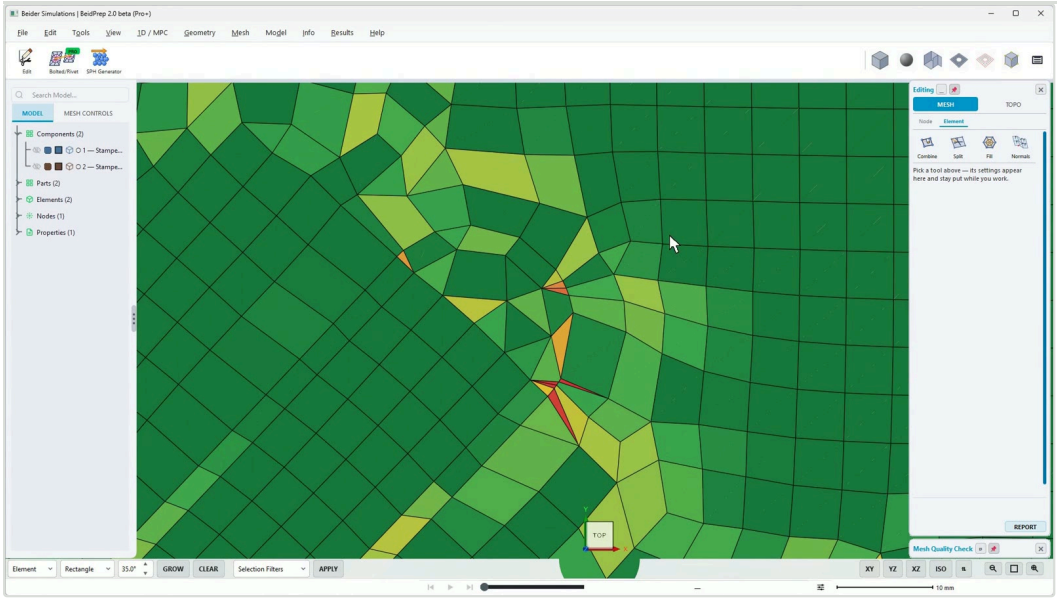
The screen at 0:12 in the video.

2 Element group

video 0:12

1. Open the Model menu, click Edit, and choose Element.

   Here, at the end of the diagonal rib, the red elements fail the limits.



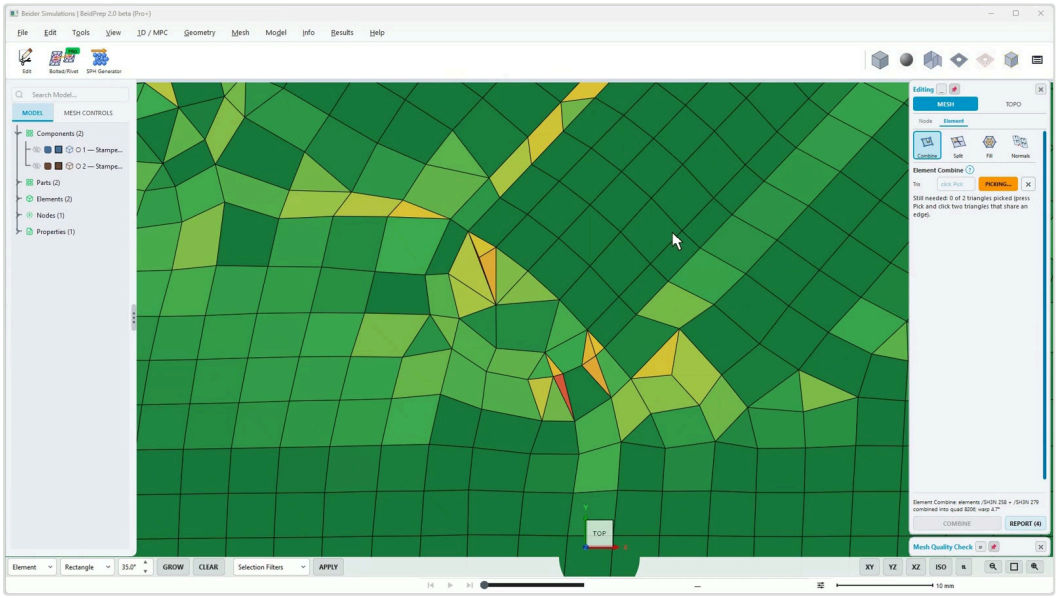
The screen at 0:22 in the video.

3 Combine

video 0:23

Combine joins two triangles into one quad.

Here two yellow triangles become one green quad: its skew drops from almost forty degrees to nine.



The screen at 0:36 in the video.

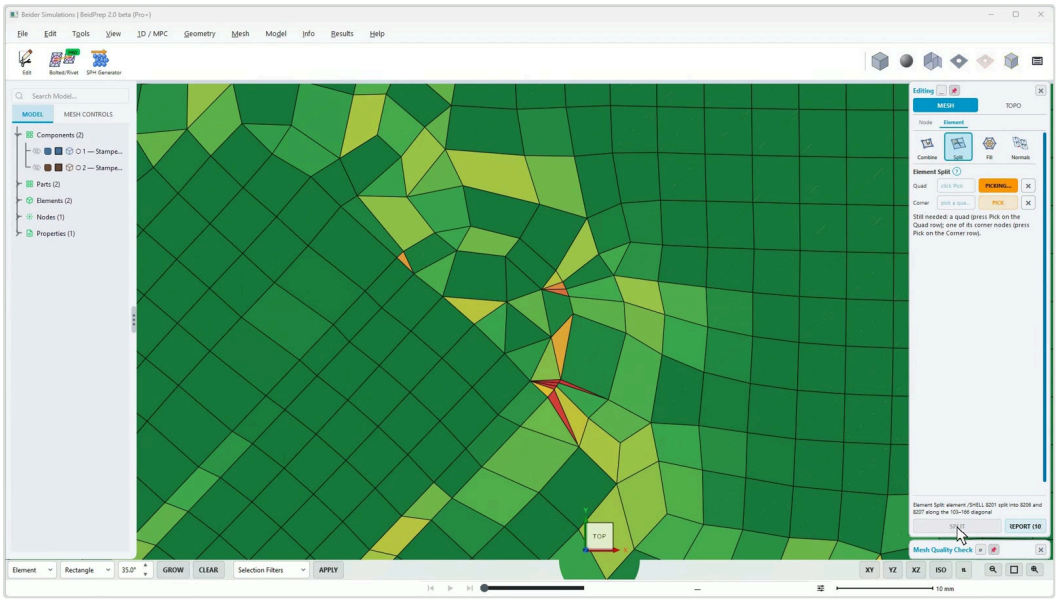
4 Split

video 0:42

Split cuts a quad into two triangles, along the diagonal from a corner you pick.

On this poor quad either diagonal leaves a sliver, so splitting it does not help.

Control Z takes it back.



The screen at 0:52 in the video.

5 Delete, Fill

video 1:00

Where several poor elements sit together, it is quicker to delete them and fill the hole.

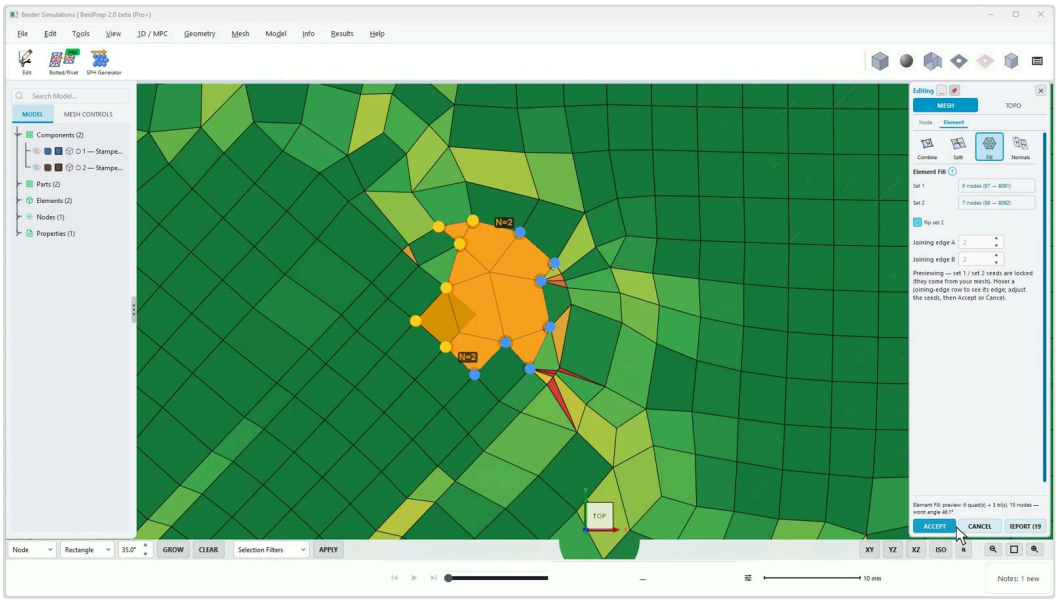
1. Drag a rectangle around them, then right click, and choose Delete.

Fill closes the hole with new elements, between two runs of nodes on its border, which are kept exactly.

Pick the two runs, set the element size to five, and Preview shows the new elements.

Accept adds them.

In the video: **5** in Elem size.



The screen at 1:25 in the video.

6 Check, The last one

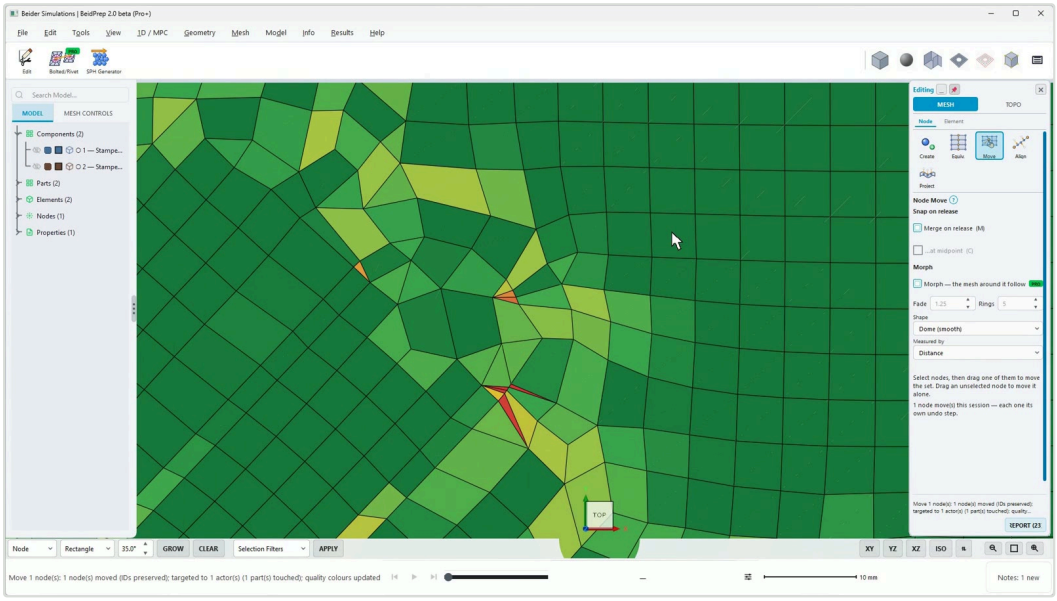
video 1:28

The new elements share every node on the border, so the mesh stays connected.

The check finds no failing element among the new ones.

This orange sliver beside the patch needs only one node moved.

1. With Move on the Node tab, drag it about three millimetres, and the sliver turns green.



The screen at 1:47 in the video.

7 Normals, Close

video 1:50

Normals turns the shell normals of a connected area to face the same way as an element you pick.

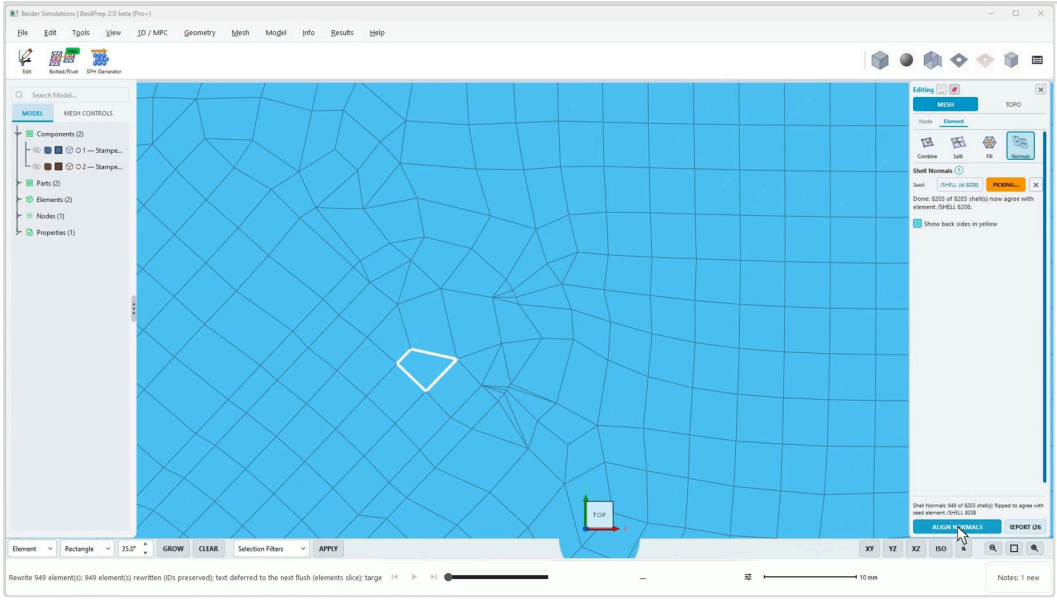
While the panel is open, a shell seen from its back is drawn yellow.

Here hundreds of the shells face the other way.

Align Normals turns them all to agree with the one you picked.

The yellow patches now show the part colour: the whole area faces the same way.

Every element edit is one step on the undo list: Control Z takes it back.



The screen at 2:06 in the video.

Steps and pictures are taken from tutorial video 06. Every picture is a frame of the video.

TUTORIAL VIDEO 07

07 Connectors: washers and rivets

Put a washer round every rivet hole of a wing bay part in one go, mesh it with a clean ring of elements round each hole, and join the bay's nine parts with rivets where their holes line up.

Tutorial files	folder `07_Connectors_washers_rivets` — open `bay_start.beid` first
Video	Watch video 07 on YouTube
Download files	Download the tutorial files (97 MB)

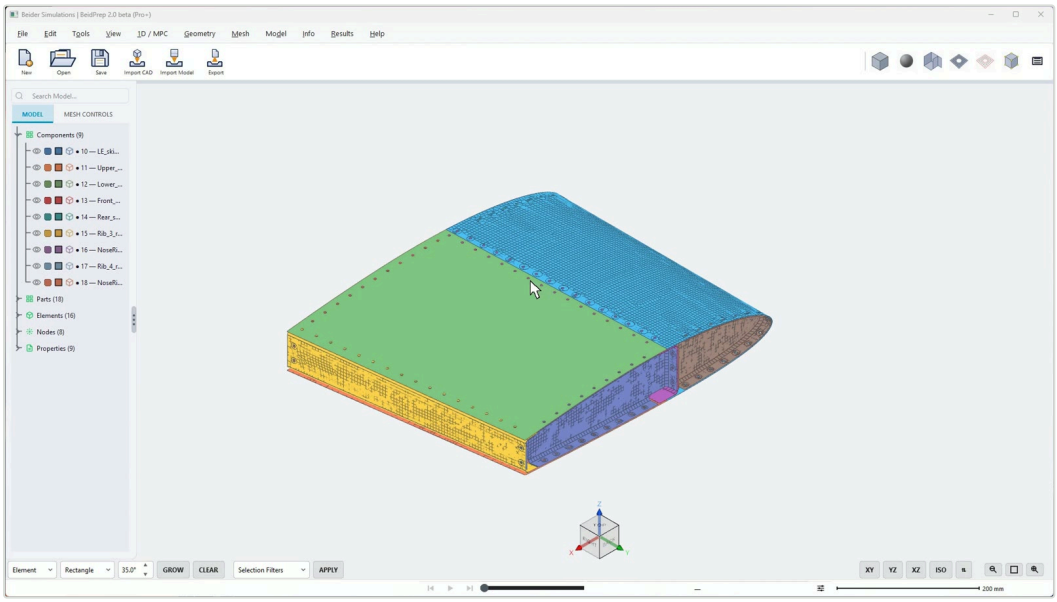
1 Opening, The bay

video 0:00

This video puts a washer round every rivet hole of a wing bay, meshes it, and joins its parts with rivets.

The bay has nine sheet-metal parts, already turned into mid-surfaces.

Eight are washered and meshed already; the upper skin is left bare, to show each step.



The screen at 0:16 in the video.

2 Washer panel, Scan the upper skin

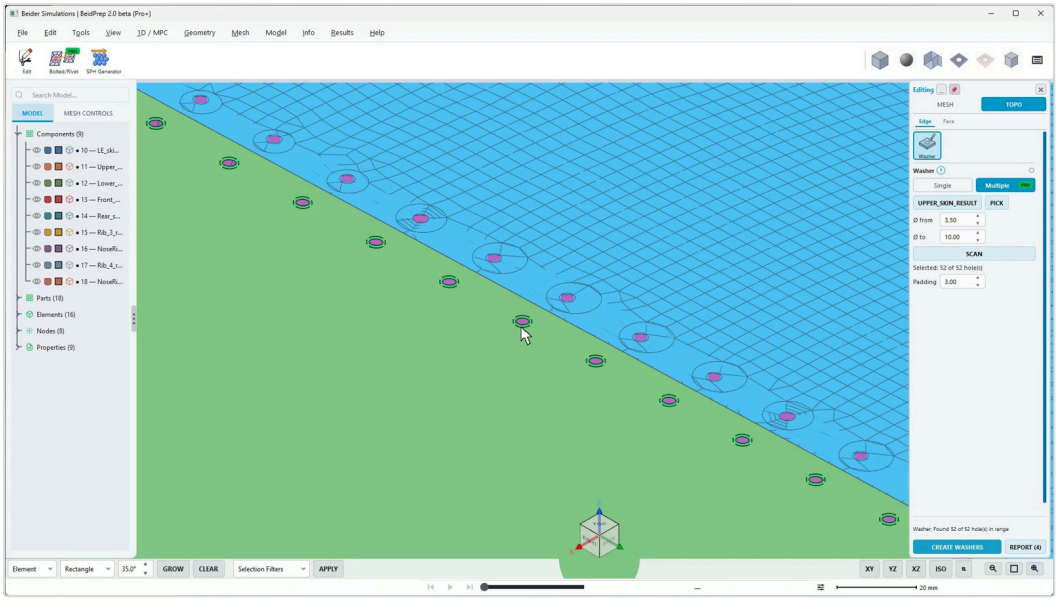
video 0:17

Washer is in the editing tools: Model, Edit, then TOPO and Edge.

1. Choose Multiple, to ring every hole of one body in one go.
2. Pick the upper skin, and set the hole sizes: from three and a half to ten millimetres.

Scan finds the holes and rings them in the view; the line under it counts them.

Control with a right click leaves a hole out and turns its ring red; Control with a left click puts it back.



The screen at 0:45 in the video.

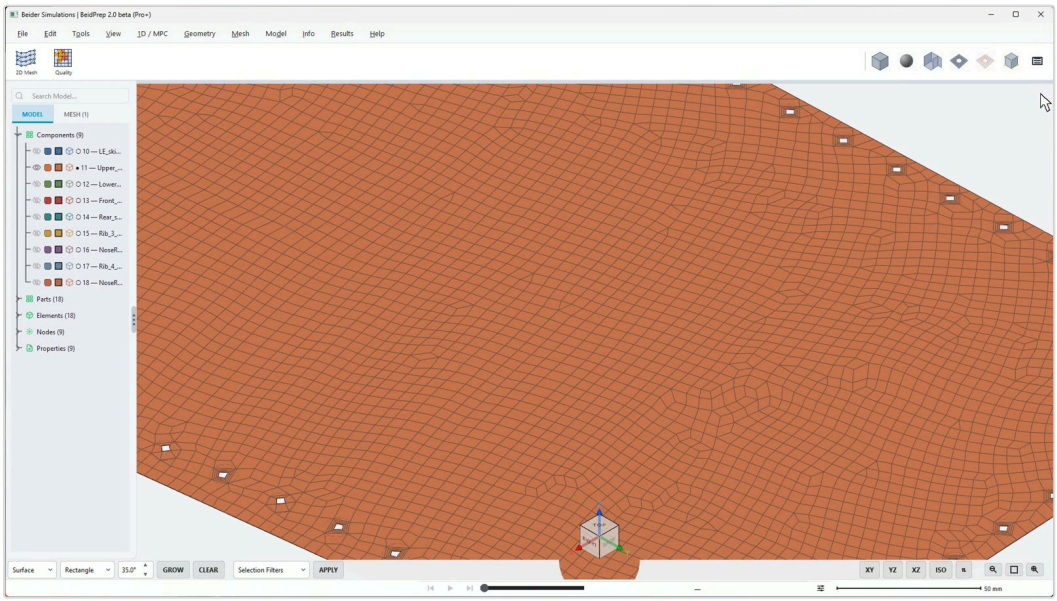
3 Create washers, Mesh the skin

video 0:45

Create washers adds a ring of faces round each hole, so the mesh gets a clean ring of elements there.

1. Show only the upper skin: right-click it in the tree, and choose Isolate.
2. Mesh it as the other parts: two D Mesh, a box round the skin, five millimetres, Quad-Dom.

   Each hole now has a clean ring of elements round it.



The screen at 1:17 in the video.

4 Bolted/Rivet

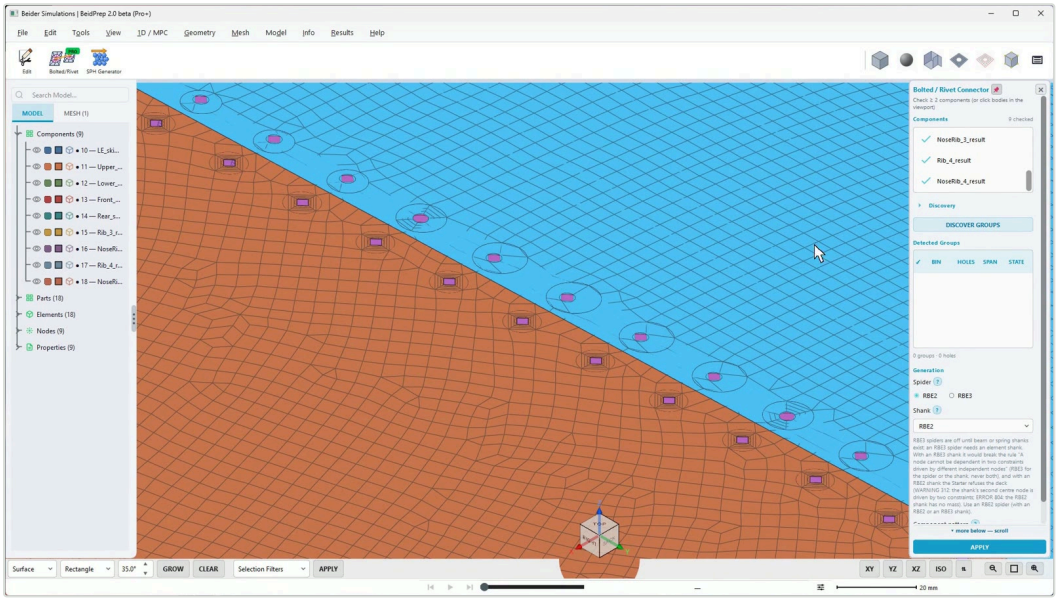
video 1:17

Show All brings the other parts back.

Bolted Rivet, on the Model tab, finds holes that line up through the parts.

1. Tick the parts to search: here, all nine.

   Discover Groups lists every line of holes it finds, with the bolt size and the span.



The screen at 1:39 in the video.

5 Apply the rivets, Close

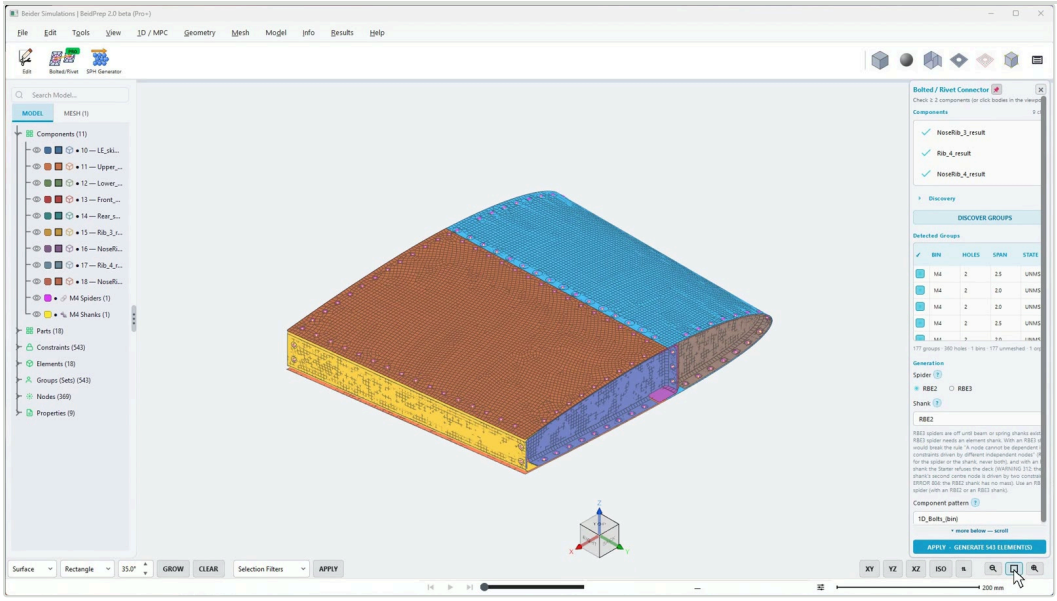
video 1:40

A spider ties each hole's edge to its centre, and a shank joins the centres: here RBE2 for both.

1. Click Apply. With this many holes, BeidPrep asks first, in case the filters are too wide.

   The progress bar counted the connectors as they were made; every rivet is now in the model, drawn on the joints.

   The bay is now one model: nine meshed parts, joined by rivets where their holes line up.



The screen at 1:54 in the video.

Steps and pictures are taken from tutorial video 07. Every picture is a frame of the video.

TUTORIAL VIDEO 08

08 Export checks and running OpenRadioss

BeidPrep checks the deck before it writes it: a broken reference is caught, then the clean deck is exported and run in OpenRadioss from inside BeidPrep, with live progress from the Engine.

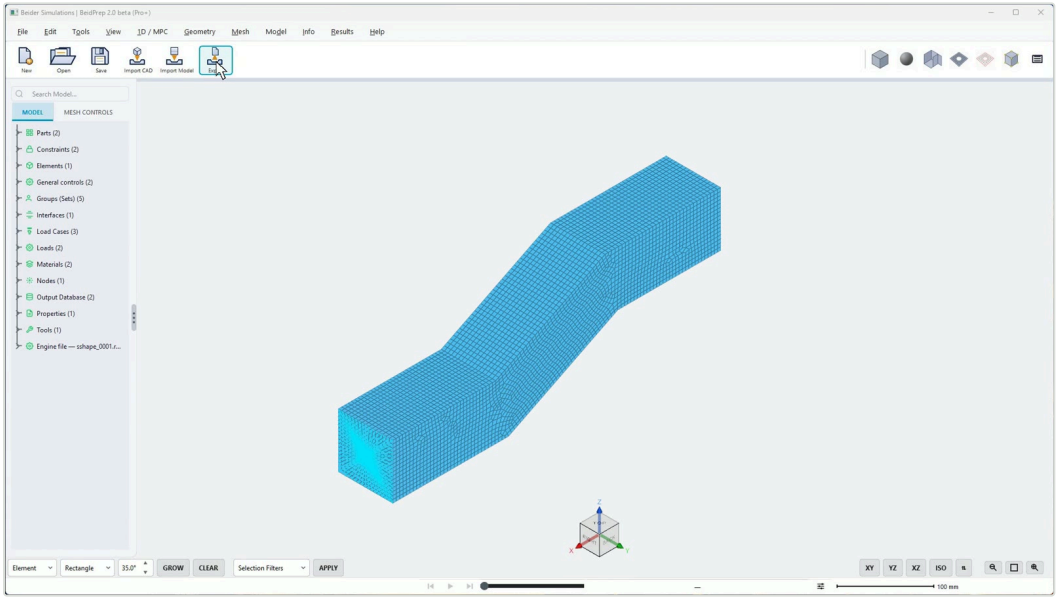
Tutorial files	folder `08_Export_checks_and_running` — open `sshape_0000.rad` first
Video	Watch video 08 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, Check before export

video 0:00

This video exports a deck, answers the check before export, and runs the model with OpenRadioss.

Before anything is written, BeidPrep checks the whole deck for what would stop the solver.



The screen at 0:11 in the video.

2 Break a reference, Export: the window

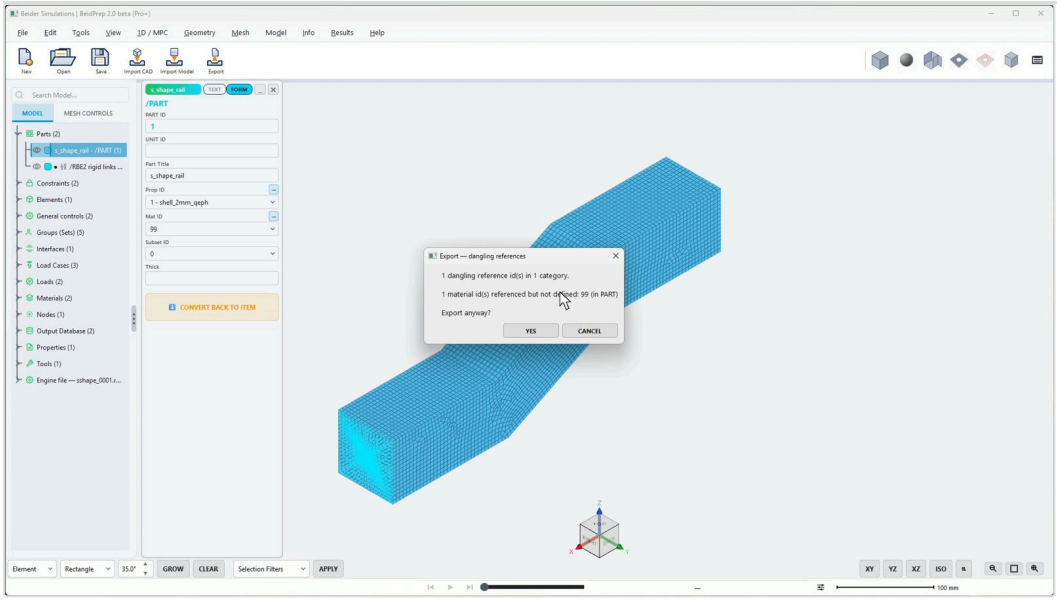
video 0:11

1. To see the check at work, point the rail's part at material ninety nine, which no card defines.
2. Click Export.

   The window says what it found: the part now names a material that no card defines.

   In the video: **99** in the /PART editor's material id field.

   In the video: **sshape** in save window: File name box.



The screen at 0:24 in the video.

3 Cancel, Undo, Export clean

video 0:25

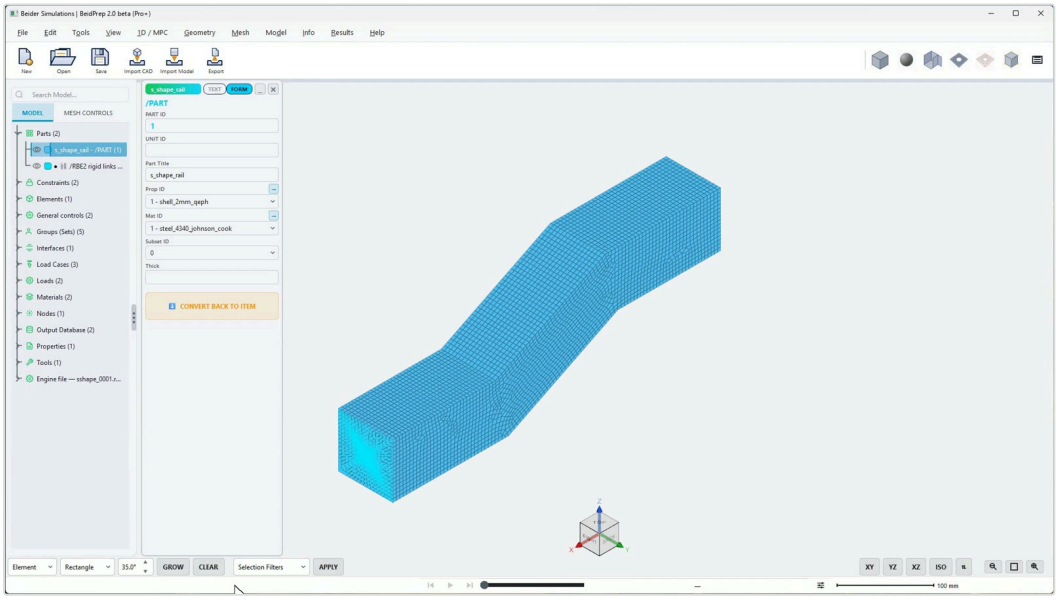
Cancel is the default, and writes nothing.

Control Z puts the part's material back.

1. Export again.

   On a clean deck the check is silent, and the status line confirms the files saved.

   In the video: **sshape** in save window: File name box.



The screen at 0:37 in the video.

4 Run analysis, Run name, Folder and threads

video 0:37

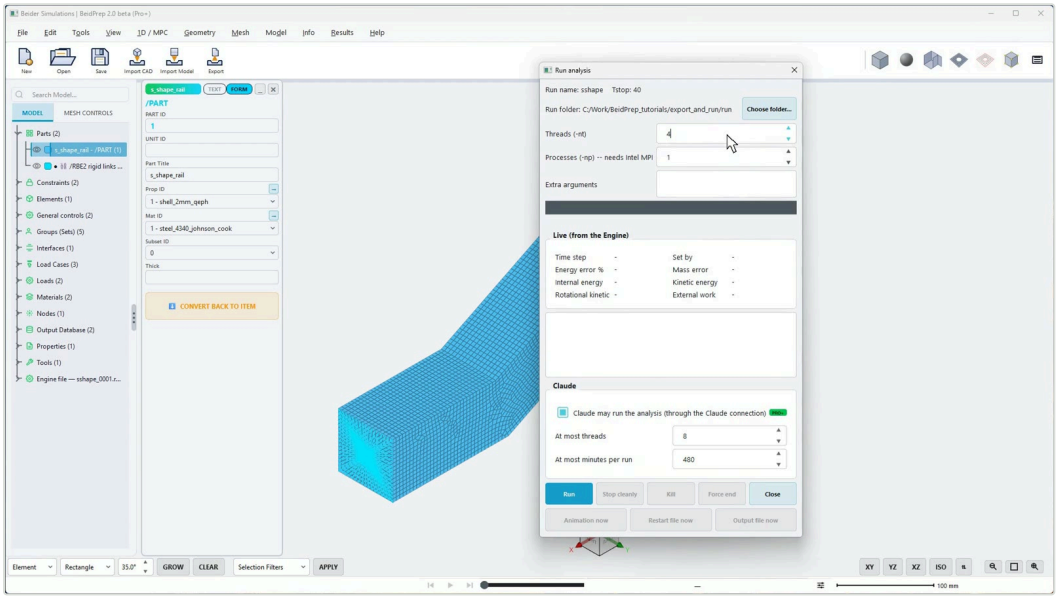
1. Now run it.

   Choose Run analysis, in the Tools menu.

   The window shows the run name and the end time, taken from the model.

2. Choose the folder to run in, and set the threads.

   In the video: **4** in Threads.



The screen at 0:50 in the video.

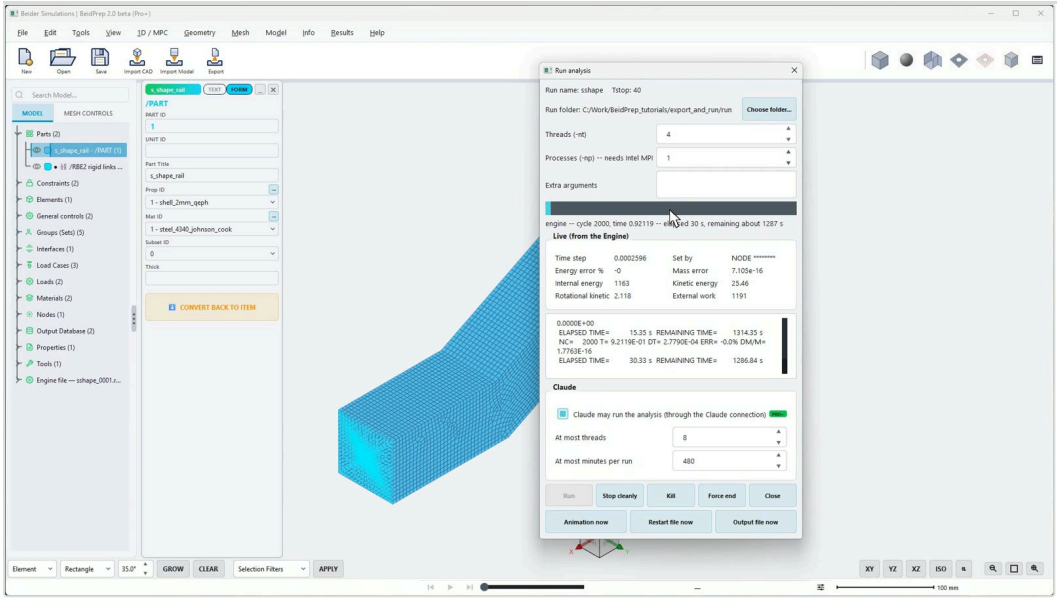
5 Run, Progress

video 0:51

1. Press Run.

   BeidPrep exports the deck with the same checks, then starts the Starter, and then the Engine.

   The progress bar shows the time reached against the end time, and the time left.



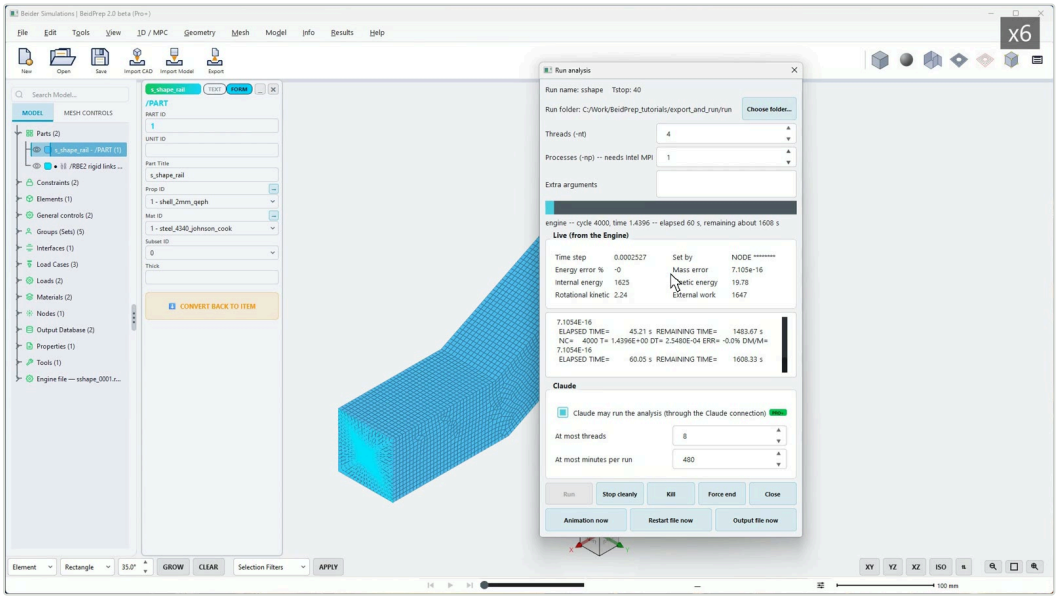
The screen at 1:04 in the video.

6 Live box, Warning signs

video 1:04

The Live box shows what the Engine itself reports: the time step, the energy error and the mass error.

A growing energy error or mass error is the first sign that a run is going wrong.



The screen at 1:15 in the video.

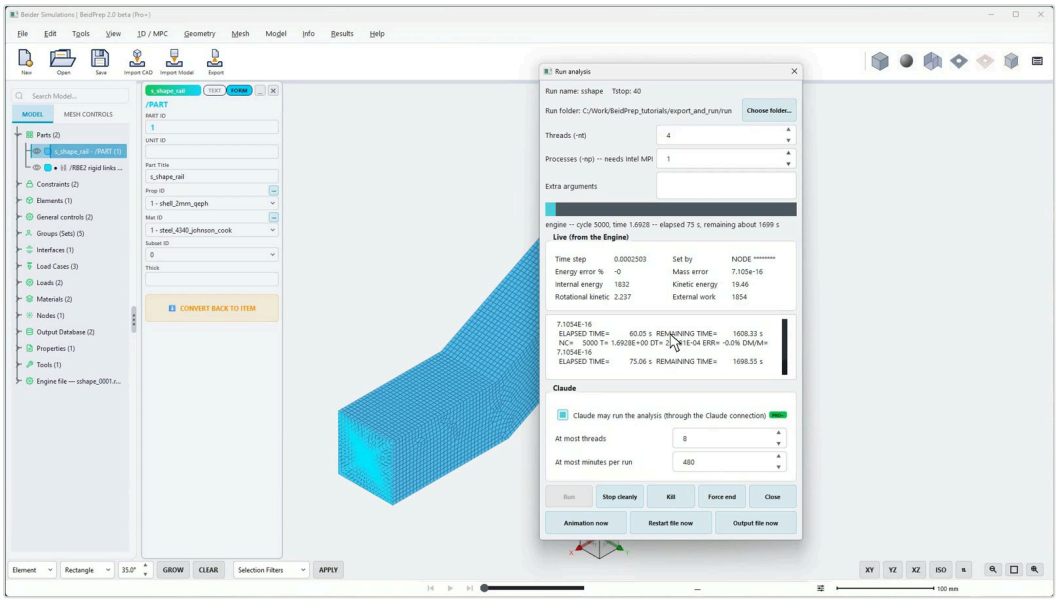
7 Stop or force end, Then results

video 1:16

Stop cleanly ends the Engine and writes a restart file.

Force end stops it at once, without one.

When the run ends, open its results, as video ten shows.



The screen at 1:26 in the video.

Steps and pictures are taken from tutorial video 08. Every picture is a frame of the video.

TUTORIAL VIDEO 09

09 From CAD to crash results: the full chain

The whole chain on one crash case: CAD to mesh, material with failure, properties, rigid ends, boundary conditions, impact velocity, contact, adaptive meshing and outputs, the run, and its results.

Tutorial files	folder `09_CAD_to_crash_results` — open `s-shape_crash.step` first (this video also needs the results zip)
Video	Watch video 09 on YouTube
Download files	Download the tutorial files (97 MB) Download the solver results (225 MB)

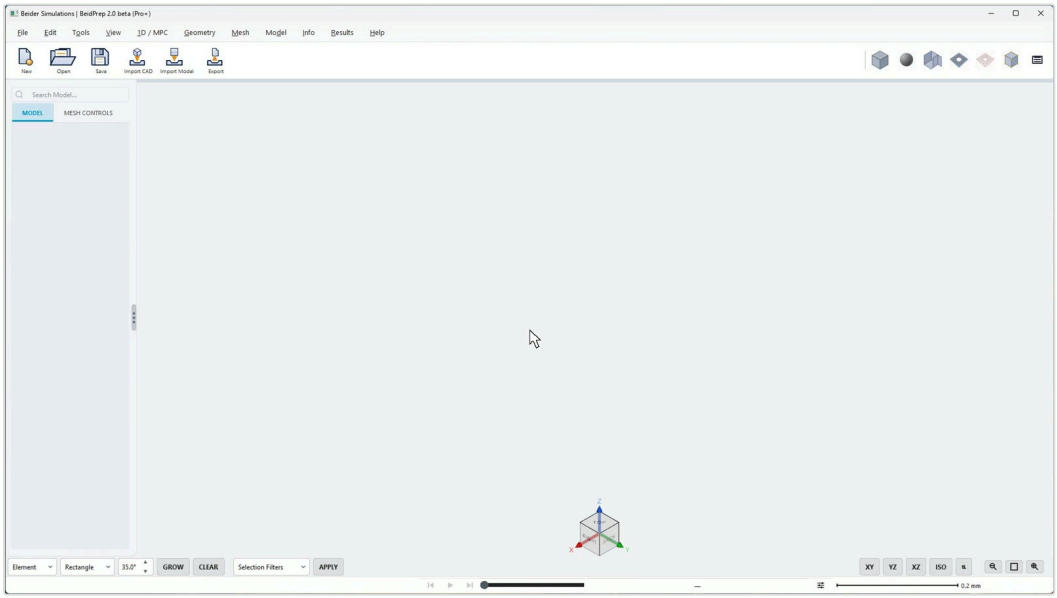
1 Opening

video 0:00

This video builds a crash model from a CAD file inside BeidPrep, runs it with OpenRadioss, and reads the results.

This is a demonstration.

The material values come from published sources, but the model has not been compared with a test.



The screen at 0:14 in the video.

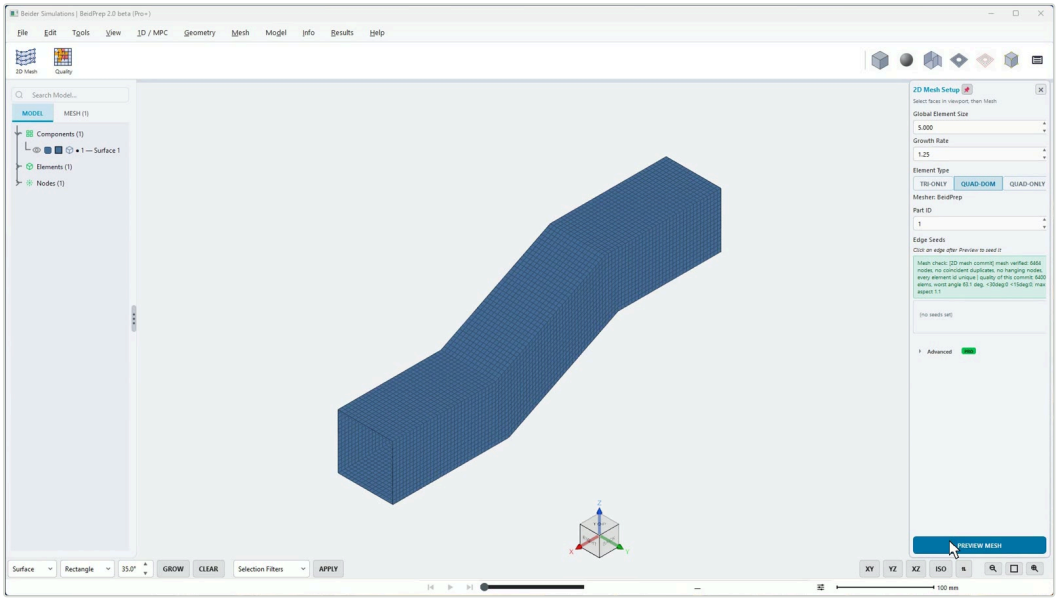
2 The case

video 0:14

The part is an S-shaped steel rail, crushed along its length between a fixed end and a moving end.

The units are kilograms, millimeters and milliseconds, so force is in kilonewtons.

The CAD file is imported, and the rail is meshed at five millimetres with Quad-Dom, as video four shows.



The screen at 0:37 in the video.

3 The mesh, Units: BEGIN

video 0:37

It must be green: connected, and of good quality.

The rail has about six thousand four hundred shells.

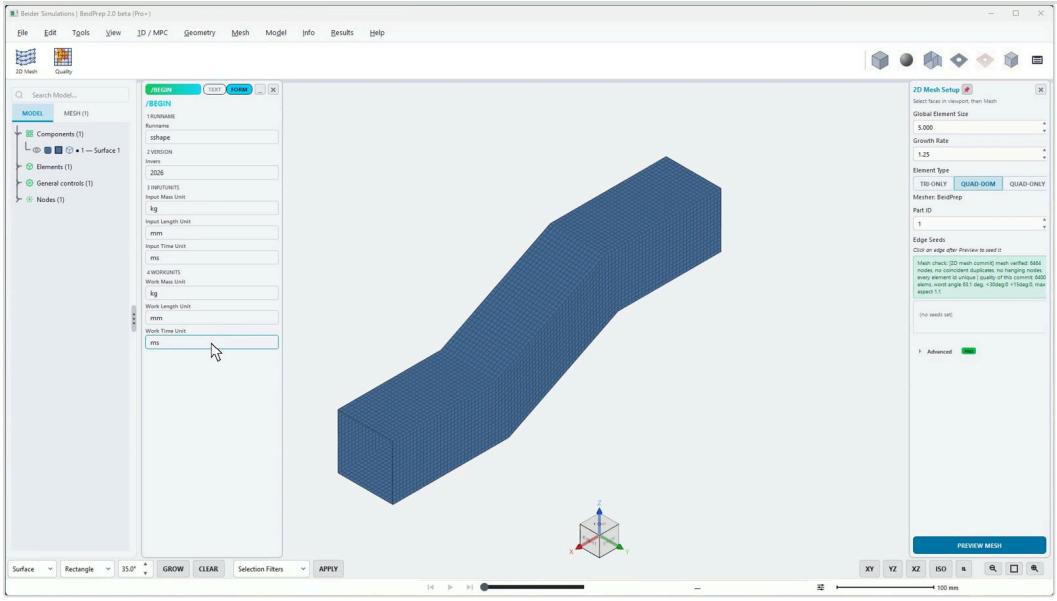
Every deck starts with its units.

1. Right-click an empty place in the model tree, and choose General controls, then BEGIN.
2. Type the run name, sshape.
3. Type the six units: kilograms, millimeters and milliseconds, for both the input and the work units.

BeidPrep fills in the input version, 2026, itself.

In the video: **sshape** in run name field.

In the video: **kg mm ms / kg mm ms** in /BEGIN unit fields.



The screen at 1:09 in the video.

4 The part

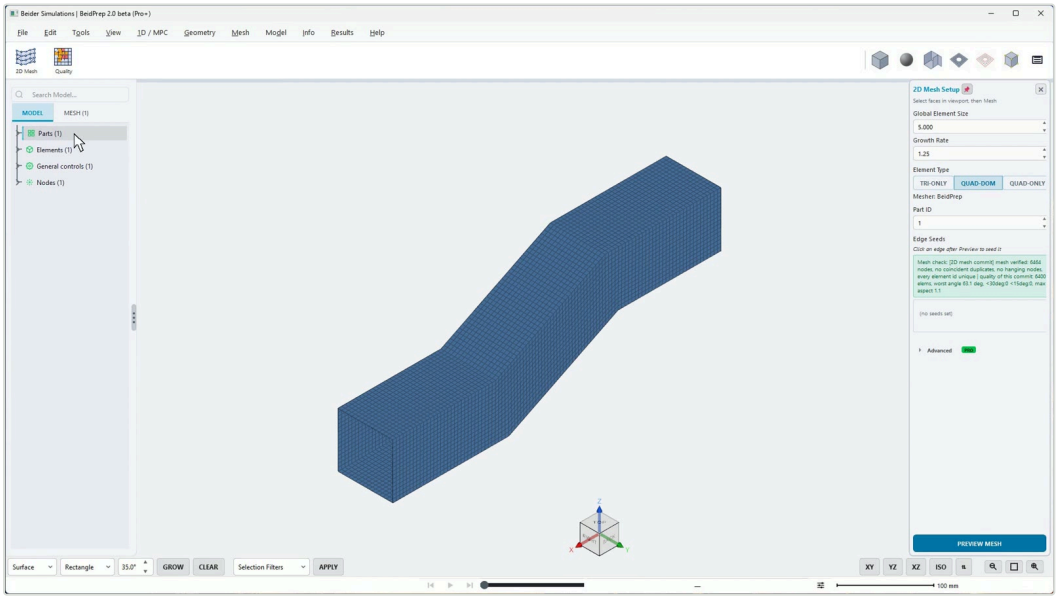
video 1:09

The shells already belong to part 1.

It is listed under Components, with the CAD's name.

1. Click it, and press CHANGE CARD TO PART.

   The tree now lists it under Parts.



The screen at 1:20 in the video.

5 Material

video 1:20

Next, the material: 4340 steel, with the Johnson-Cook material law.

1. Right-click the tree, and choose Materials, Elasto-Plastic, MAT LAW 2.
2. Type a title, then the values on screen: density, Young's modulus in gigapascals, Poisson's ratio, the Johnson-Cook a, b, n and c, and a reference strain rate of one per second, written per millisecond.

   In the video: **steel 4340 Johnson-Cook** in field: /MAT/LAW2 mat_title.

   In the video: **7.83e-6** in field: /MAT/LAW2 rho_i.

   In the video: **208** in field: /MAT/LAW2 E.

   In the video: **0.3** in field: /MAT/LAW2 nu.

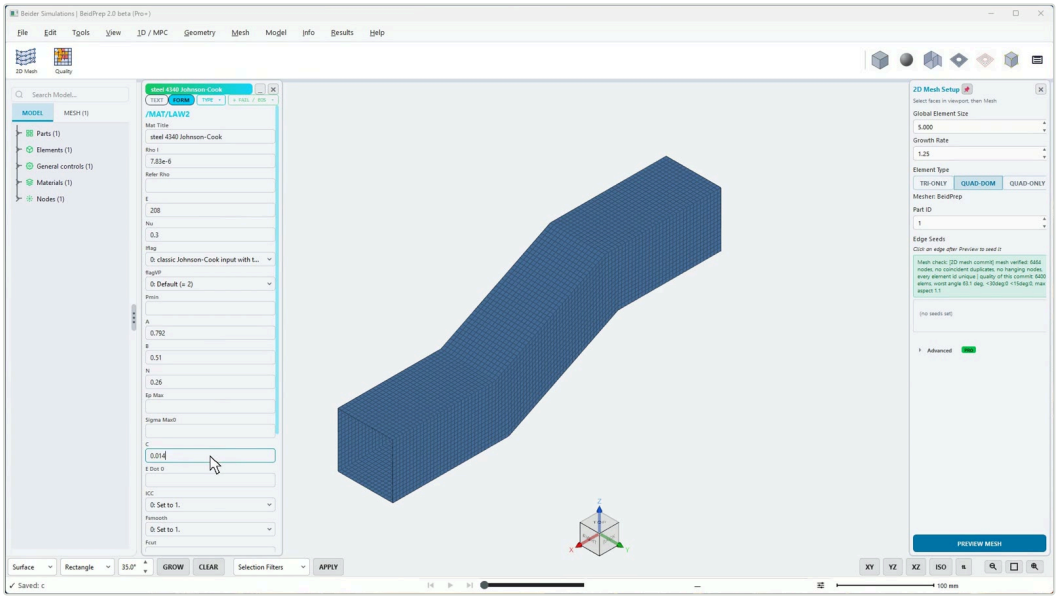
   In the video: **0.792** in field: /MAT/LAW2 a.

   In the video: **0.51** in field: /MAT/LAW2 b.

   In the video: **0.26** in field: /MAT/LAW2 n.

   In the video: **0.014** in field: /MAT/LAW2 c.

   In the video: **0.001** in field: /MAT/LAW2 e_dot_0.



The screen at 1:47 in the video.

6 Failure

video 1:47

1. On the material's form, press plus FAIL, EOS.
2. Choose Add failure model, then FAIL JOHNSON.

   The status line says it was created for material 1.
3. Type D1, 0.05; D2, 3.44; and D3, minus 2.12.
4. Leave D4 and D5 blank.

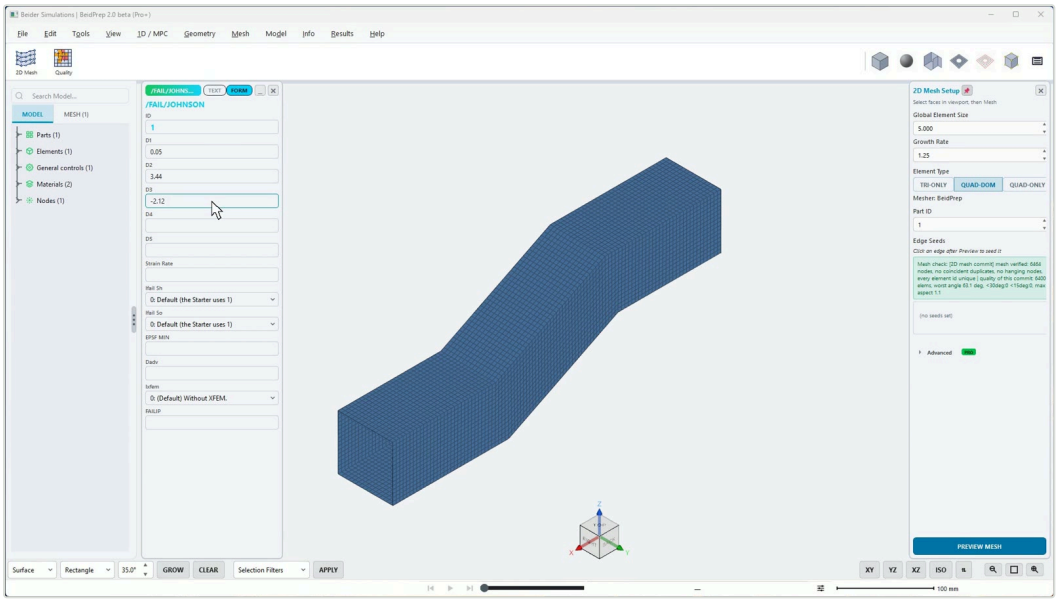
   They are zero: no rate or temperature effect on failure.

   An element is deleted when its accumulated damage reaches one.

In the video: **0.05** in field: /FAIL/JOHNSON D1.

In the video: **3.44** in field: /FAIL/JOHNSON D2.

In the video: **-2.12** in field: /FAIL/JOHNSON D3.



The screen at 2:18 in the video.

7 Shell property

video 2:18

Now the shell property.

1. Right-click the tree, and choose Properties, Shell, PROP TYPE 1.
2. Set Ishell to 24: physical hourglass control, with no artificial stiffness.
3. Set N to 5 points through the thickness, and Thick to 2 millimeters.
4. Set Ithick to 1, so the shells thin as they stretch.
5. Set Iplas to 1.

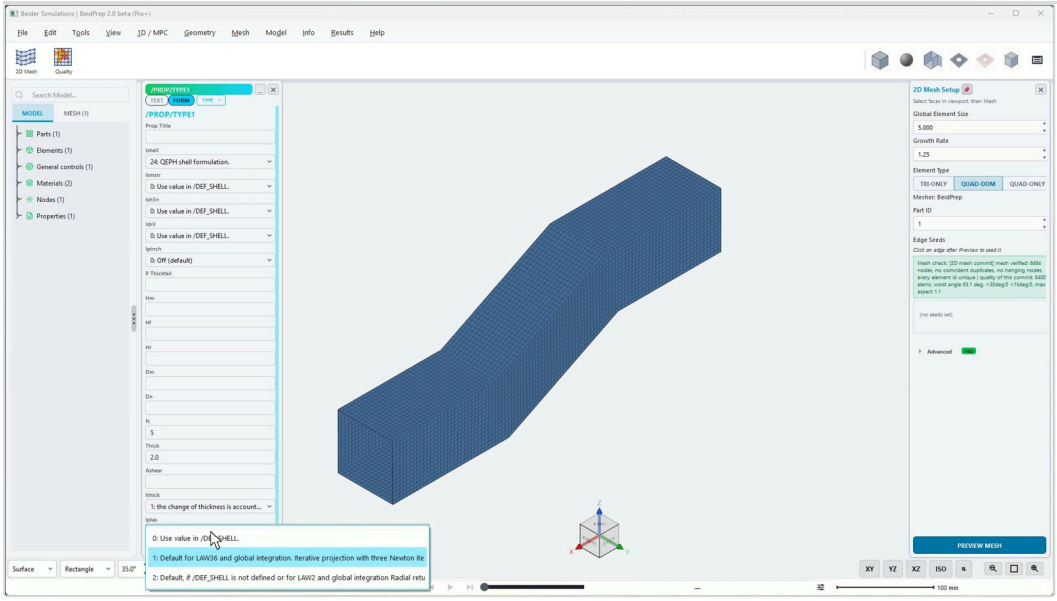
In the video: **24** in flag: /PROP/TYPE1 Ishell.

In the video: **5** in field: /PROP/TYPE1 N.

In the video: **2.0** in field: /PROP/TYPE1 Thick.

In the video: **1** in flag: /PROP/TYPE1 Ithick.

In the video: **1** in flag: /PROP/TYPE1 Iplas.



The screen at 2:42 in the video.

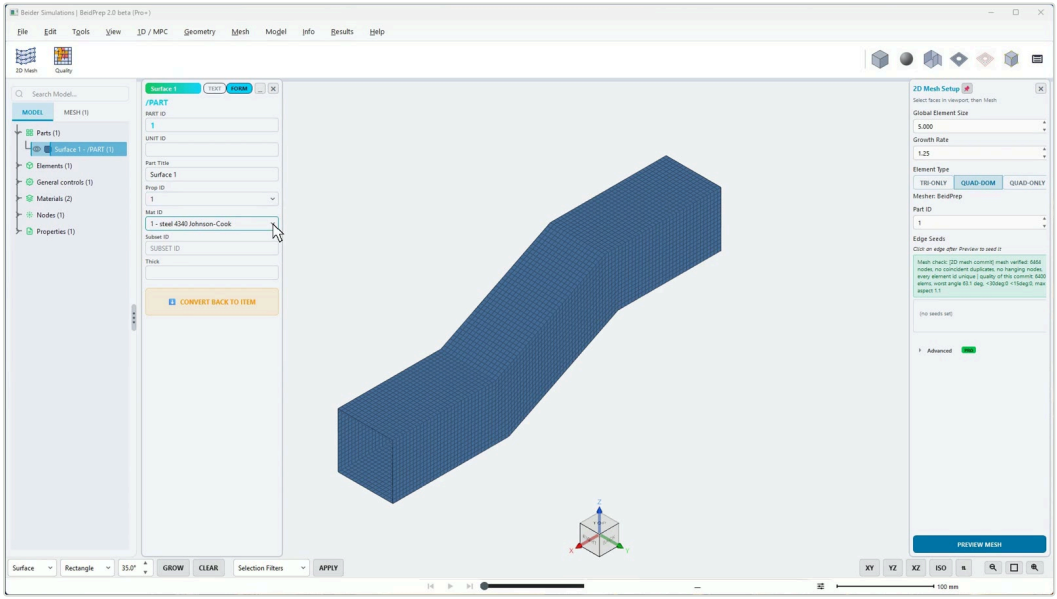
8 Part ids

video 2:43

1. Give the part this property and the material.
2. Click the part under Parts.
3. Choose property 1 in the Prop ID list, and material 1 in the Mat ID list.

   In the video: **1** in list: /PART prop_ID.

   In the video: **1** in list: /PART mat_ID.



The screen at 2:53 in the video.

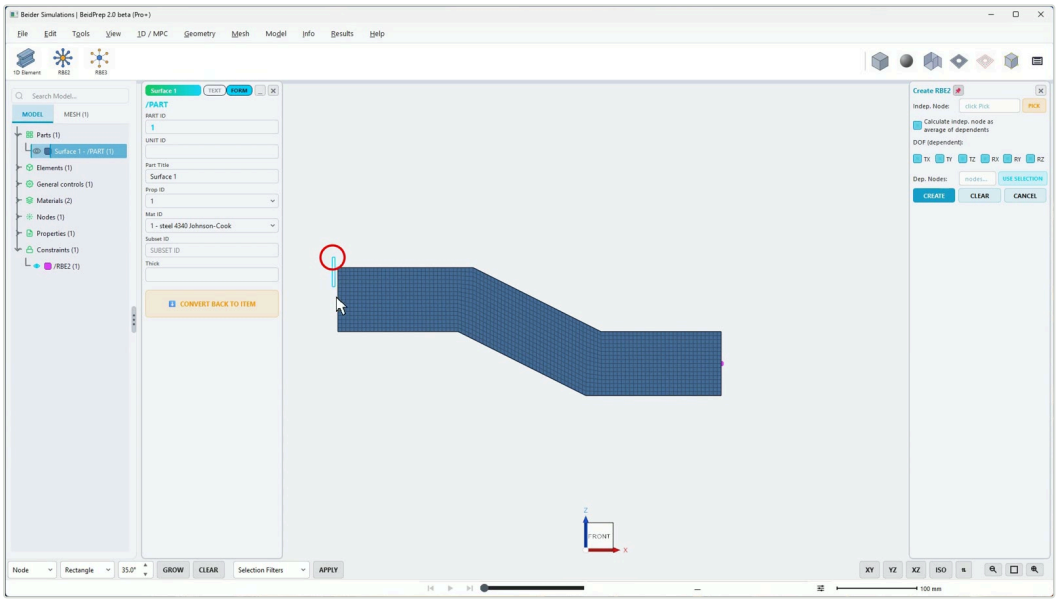
9 RBE2 ends

video 2:53

Now the two ends.

1. Click RBE2 on the 1D MPC tab.
2. Tick Calculate independent node as average of dependents.
3. Drag a rectangle around the end at the largest x.
4. Press USE SELECTION, then CREATE.

   The status line names the main node, made at the middle of the end.
5. Do the same at the smallest x.



The screen at 3:17 in the video.

10 Node groups

video 3:22

The RBE2 group holds the tied nodes, not the main node.

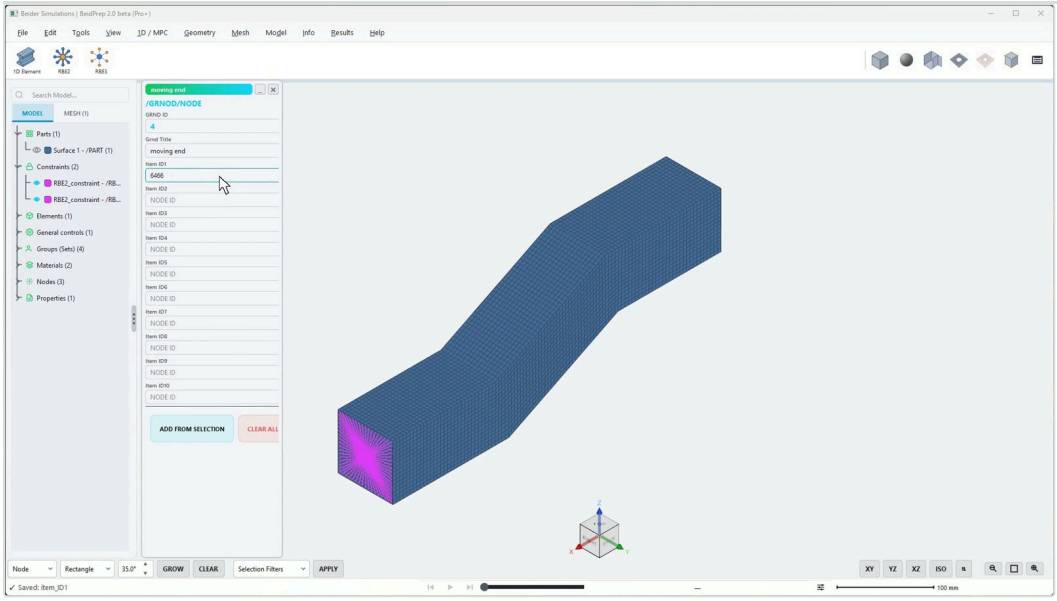
1. So make one node group per end.
2. Press ISO at the bottom right.
3. Right-click the tree, and choose Groups, then GRNOD NODE.
4. Type the title, fixed end, and the end's main node number in Item ID1.

   In the video: **fixed end** in field: /GRNOD/NODE grnd_title.

   In the video: **6465** in field: /GRNOD/NODE item_ID1.

   In the video: **moving end** in field: /GRNOD/NODE grnd_title.

   In the video: **6466** in field: /GRNOD/NODE item_ID1.



The screen at 3:48 in the video.

11 Fixed end

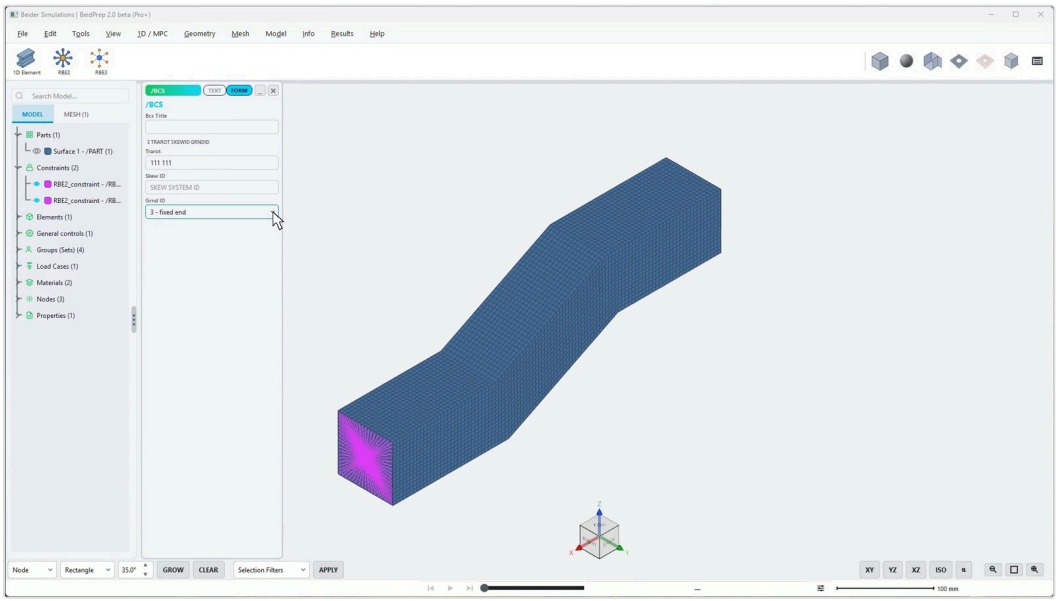
video 3:48

The fixed end.

1. Right-click the tree, and choose Load Cases, Boundary Conditions, BCS.
2. Fix all six directions.
3. Choose the fixed end's main-node group in Grnd ID.

   In the video: **111111** in field: /BCS Trarot.

   In the video: **3** in list: /BCS grnd_ID.



The screen at 4:03 in the video.

12 Moving end

video 4:04

The moving end needs three cards.

First, a BCS on its main-node group that leaves only x free, the other five directions fixed.

Second, a function, FUNCT, under Tools, that is 1 from time 0 to 1000.

Third, an IMPVEL, under Load Cases, Imposed Motion.

1. Choose that group and the function from their lists, the direction X, and a scale of 6 millimeters per millisecond.

In the video: **011111** in field: /BCS Trarot.

In the video: **4** in list: /BCS grnd_ID.

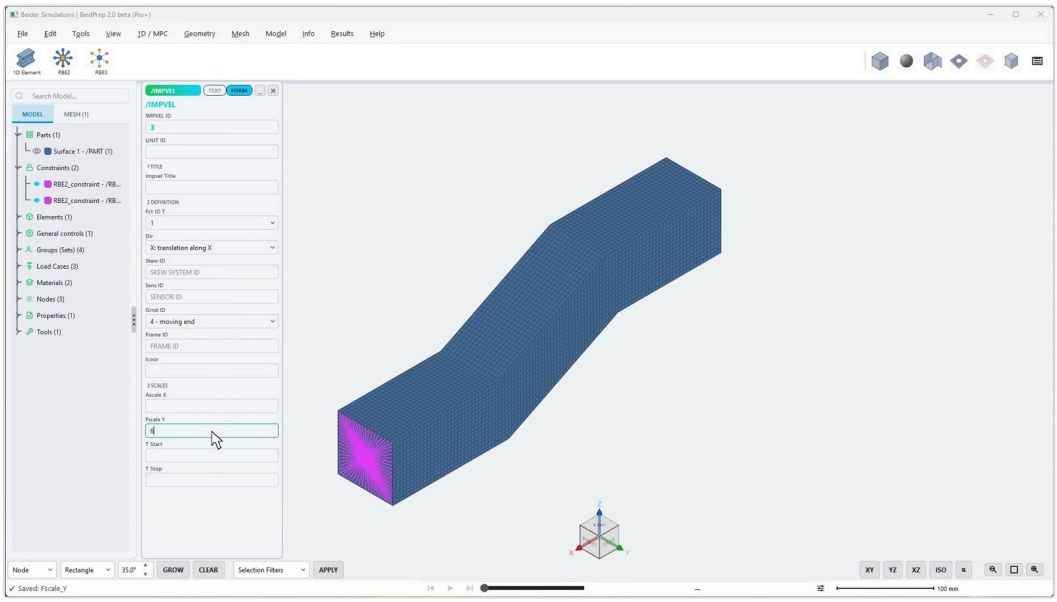
In the video: **0,1 1000,1** in points: /FUNCT.

In the video: **4** in list: /IMPVEL grnd_ID.

In the video: **1** in list: /IMPVEL fct_ID_T.

In the video: **X** in flag: /IMPVEL Dir.

In the video: **6** in field: /IMPVEL Fscale_Y.



The screen at 4:43 in the video.

13 Self-contact

video 4:43

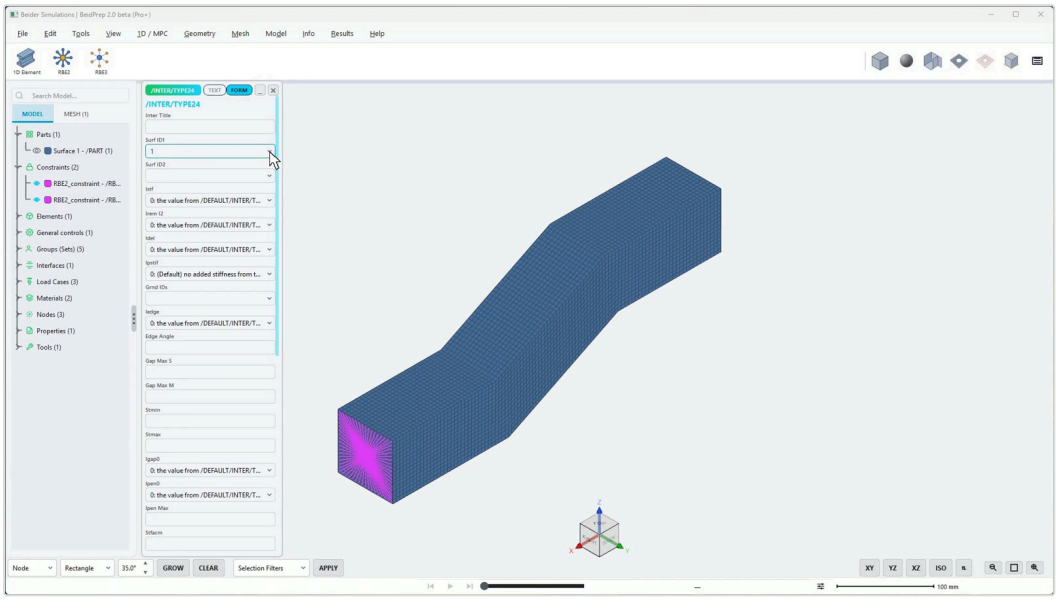
As the rail folds, its walls meet.

The contact needs a surface, so make one first: right-click the tree, create SURF PART, and choose the rail from the list.

1. Then right-click the tree, and choose Interfaces, Penalty Contact, INTER TYPE 24.
2. Choose that surface in the Surf ID1 list, so the walls cannot pass through each other.

In the video: **1** in list: /SURF/PART item_ID1.

In the video: **1** in list: /INTER/TYPE24 surf_ID1.



The screen at 5:07 in the video.

14 Adaptive meshing

video 5:08

Adaptive meshing lets OpenRadioss split a shell into four smaller ones during the run, where the result needs it.

1. Create ADMESH GLOBAL with one level.
2. Create ADMESH SET, choose the rail from the list, and set a thickness error of 0.04 millimeters, and the angle criterion at 179.9 degrees.

   Then only thinning, not bending, splits a shell.

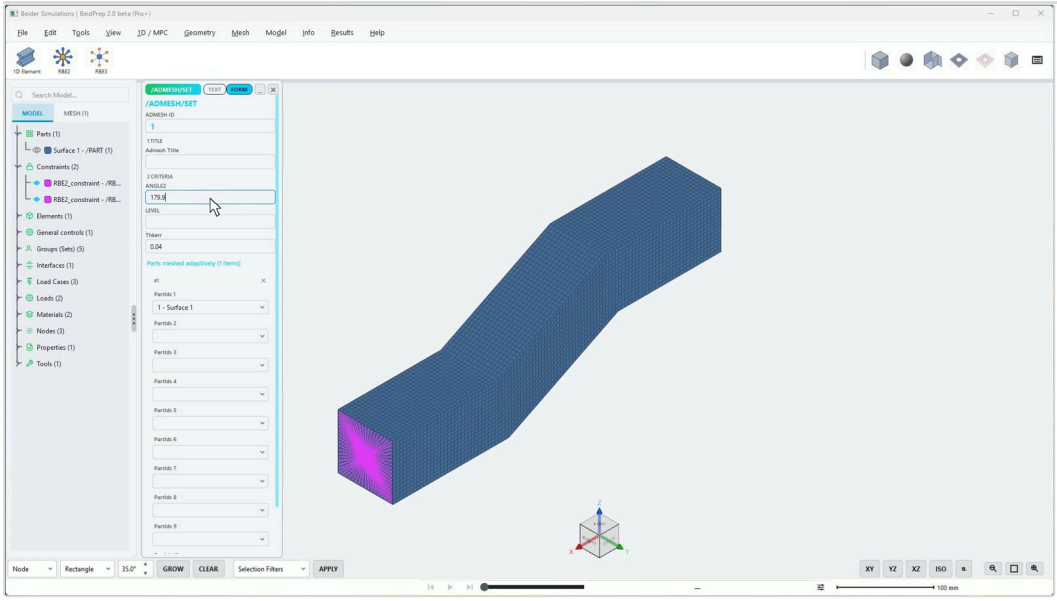
   One level is plenty here: the contact is searched on every refined shell, which slows the run sharply.

   In the video: **1** in field: /ADMESH/GLOBAL LEVEL.

   In the video: **1** in list: /ADMESH/SET PartIds_1.

   In the video: **0.04** in field: /ADMESH/SET Thkerr.

   In the video: **179.9** in field: /ADMESH/SET ANGLE2.



The screen at 5:43 in the video.

15 Run settings

video 5:44

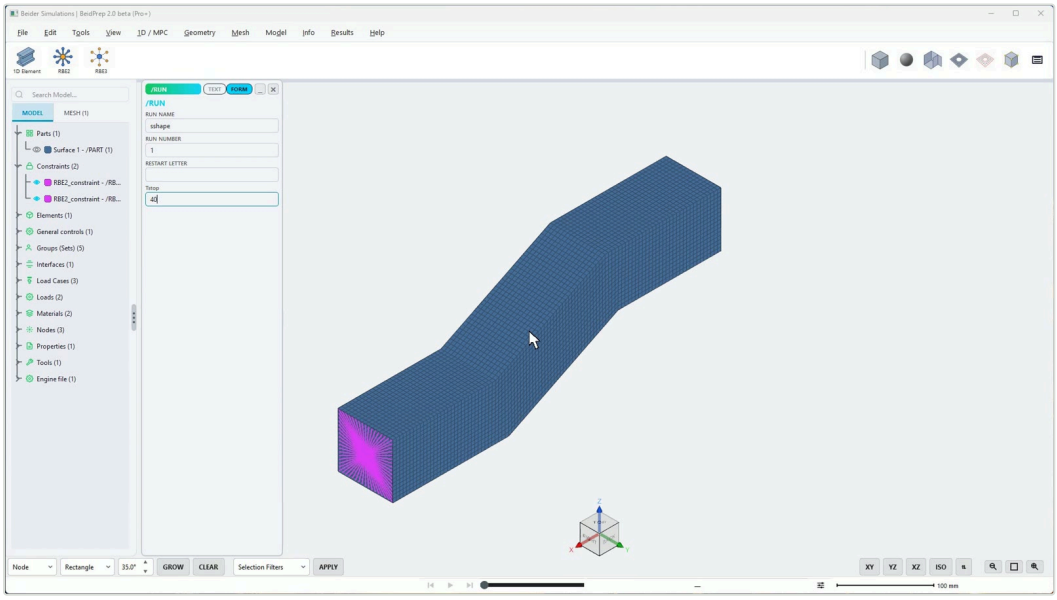
The Engine file holds how the run goes.

1. Create RUN, under General controls, with an end time of 40 milliseconds.

   The brief asks for no mass scaling, so the solver keeps its own stable time step.

   As adaptive meshing refines the shells, that step gets shorter, and the run takes longer.

   In the video: **40** in field: /RUN Tstop.



The screen at 6:02 in the video.

16 Outputs

video 6:02

Now the outputs.

ANIM DT, starting at 0, every 2 milliseconds.

Then the animation results: plastic strain, von Mises stress, the shell stress and strain tensors, thickness, and displacement.

TFILE, every 0.1 milliseconds.

TH NODE on the fixed end's main node, with the reaction force.

And TH PART, with the rail chosen from the list, and its internal, kinetic and hourglass energy.

The run is started from the Tools menu, as video eight shows.

In the video: **0** in field: /ANIM/DT Tstart.

In the video: **2** in field: /ANIM/DT Tfreq.

In the video: **0.1** in field: /TFILE dT.

In the video: **REACX** in field: /TH/NODE var_ID_1.

In the video: **REACY** in field: /TH/NODE var_ID_2.

In the video: **REACZ** in field: /TH/NODE var_ID_3.

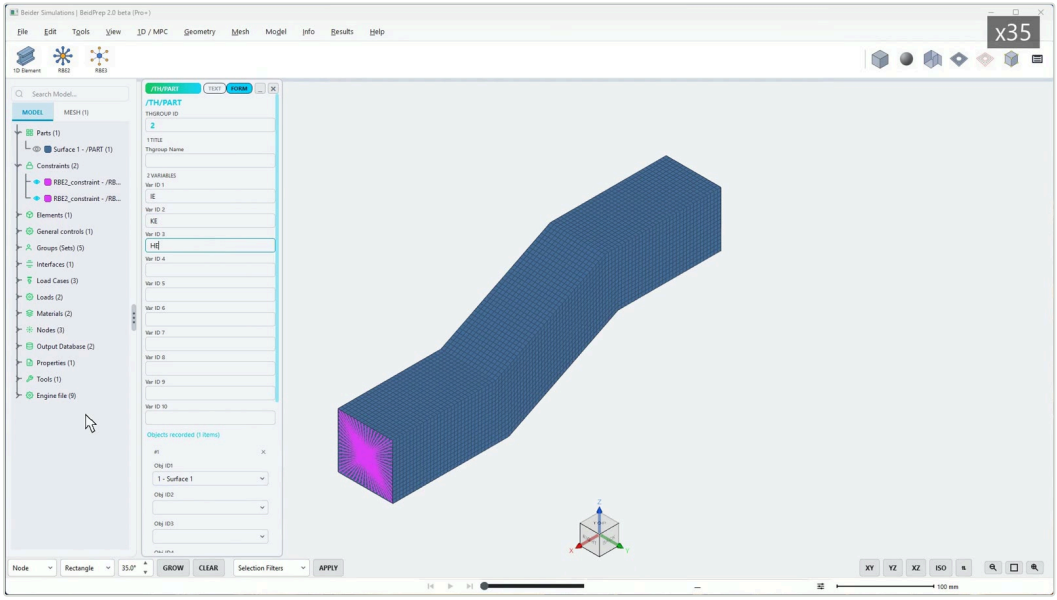
In the video: **6465** in field: /TH/NODE node_ID.

In the video: **1** in list: /TH/PART Obj_ID1.

In the video: **IE** in field: /TH/PART var_ID_1.

In the video: **KE** in field: /TH/PART var_ID_2.

In the video: **HE** in field: /TH/PART var_ID_3.



The screen at 7:29 in the video.

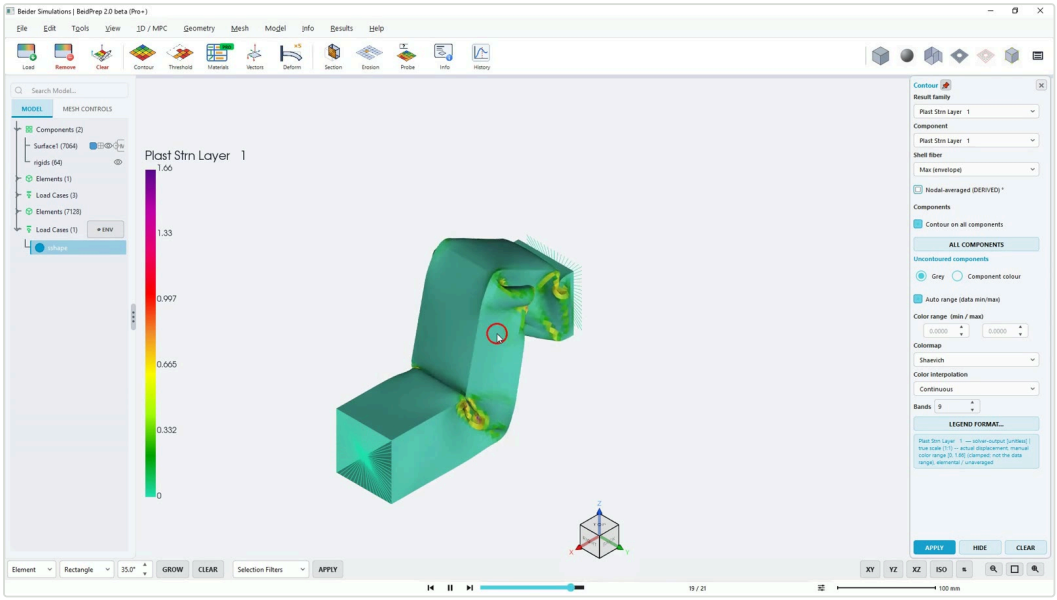
17 The run

video 7:36

With no mass scaling, no mass is added, so the mass error stays at zero.

The run takes a while, so the video skips ahead to its end.

The run's results are opened and coloured by plastic strain, as video ten shows.



The screen at 8:01 in the video.

18 Plastic strain, Refined shells

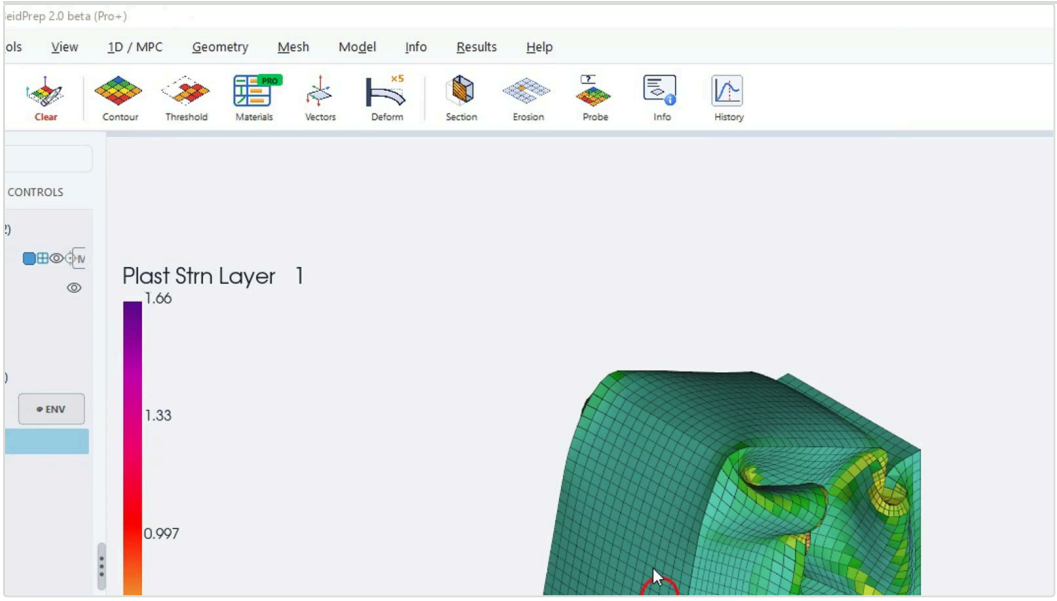
video 8:01

At the end, the folds carry the highest plastic strain, and the straight walls stay almost elastic.

1. Click the grid icon on the rail's row in the tree, to turn its mesh lines on.

The smaller shells in the folds were split during the run.

By the end, the rail has about 670 more shells than it started with.



The screen at 8:19 in the video.

19 Failed shells, Reaction force

video 8:20

1. Click Erosion, and choose Hide.

   Holes are left where shells failed and were removed, in the sharpest folds.

2. Click History on the Results tab.

3. Under Nodes, tick the fixed end's REACX.

   This is the force the fixed end takes as the rail crushes.

   It averages about 85 kilonewtons.

   It is drawn below zero, because it pushes against the x direction.



The screen at 8:42 in the video.

20 Energies

video 8:48

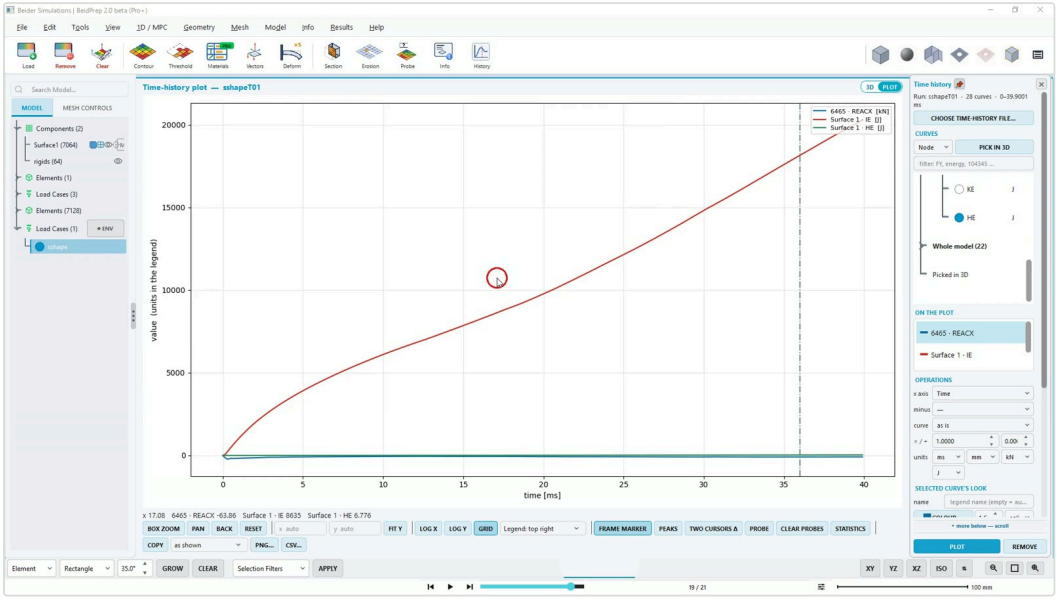
The hourglass energy should stay small against the internal energy.

Here it ends at 0.16 percent.

As a check, about 20,300 joules over 240 millimeters of crush gives the same 85 kilonewtons.

This is a demonstration, not a design result.

Model: Beider Simulations.



The screen at 9:09 in the video.

Steps and pictures are taken from tutorial video 09. Every picture is a frame of the video.

TUTORIAL VIDEO 10

10 Results: contour, deform, section, threshold, probe

Read crash results in BeidPrep: contour plots, playback, the deformed shape, sections, thresholds, displacement vectors, probing a single element and the model summary.

Tutorial files	folder `10_Results` — open `sshape_0000.rad` first (this video also needs the results zip)
Video	Watch video 10 on YouTube
Download files	Download the tutorial files (97 MB) Download the solver results (225 MB)

1 Opening, Load

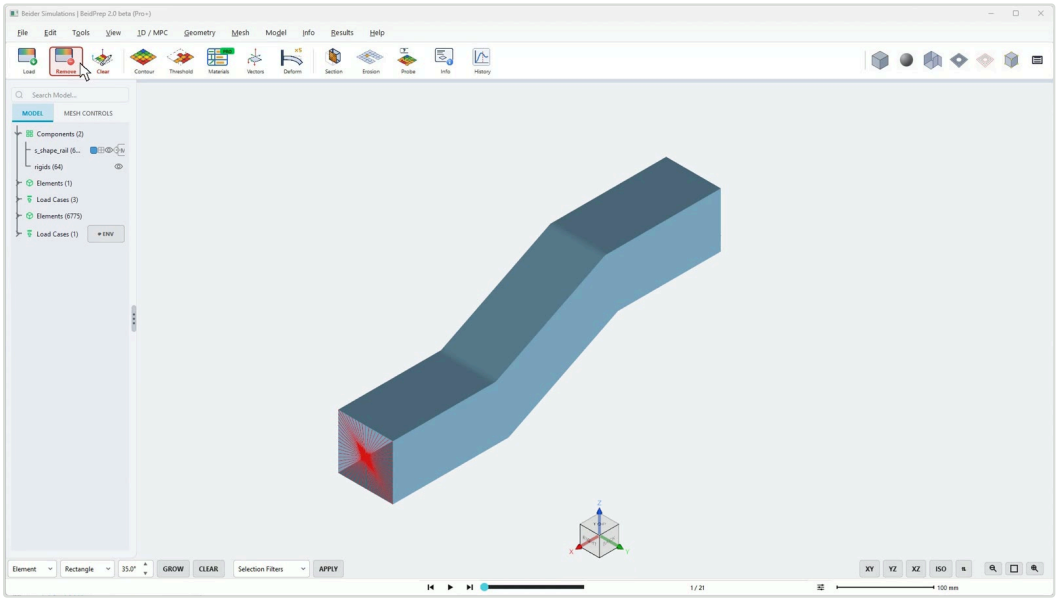
video 0:00

This video shows the results of a crash run in BeidPrep.

The S-shaped rail is crushed at six metres per second.

Load has already opened the run's first animation file.

Each file is one moment of the run.



The screen at 0:13 in the video.

2 Load case, Contour

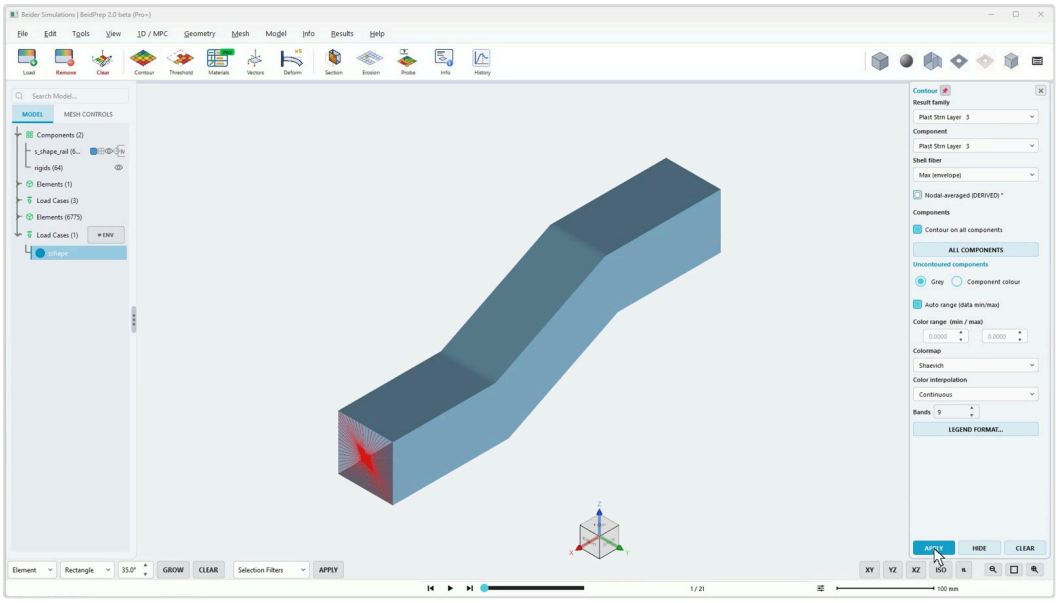
video 0:13

1. In the model tree, click the run's load case.

   The result tools need it.

2. Click Contour, and choose Plastic Strain.

   Nothing changes in the 3D view, until you press Apply.



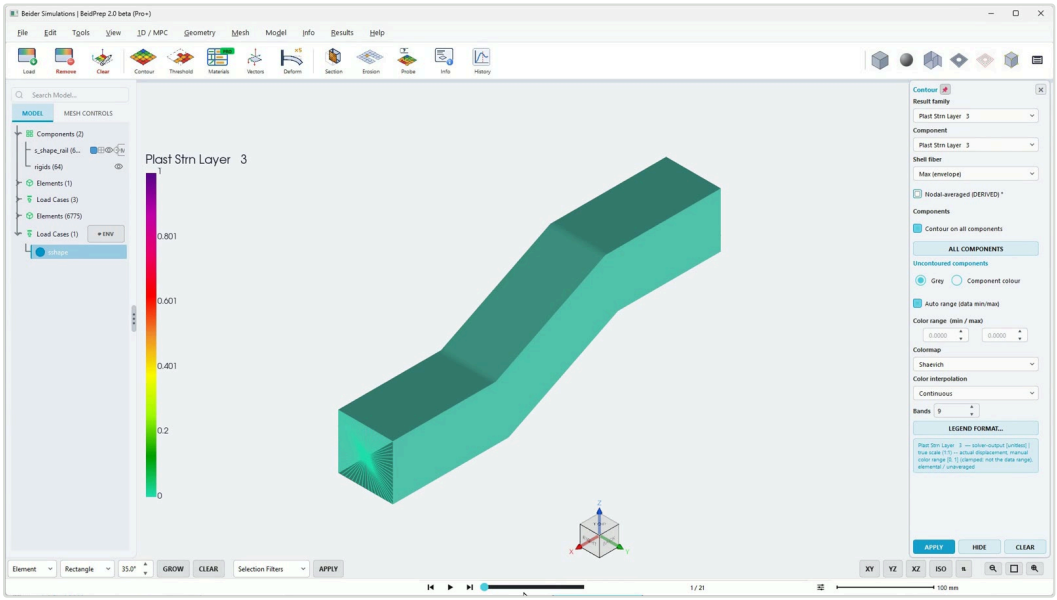
The screen at 0:25 in the video.

3 Apply rule, Playback bar

video 0:25

1. Press Apply: at this first moment of the run, the plastic strain is still zero everywhere.
2. Now play the run.

   The playback controls are on the bottom bar.



The screen at 0:36 in the video.

4 Play, Slider

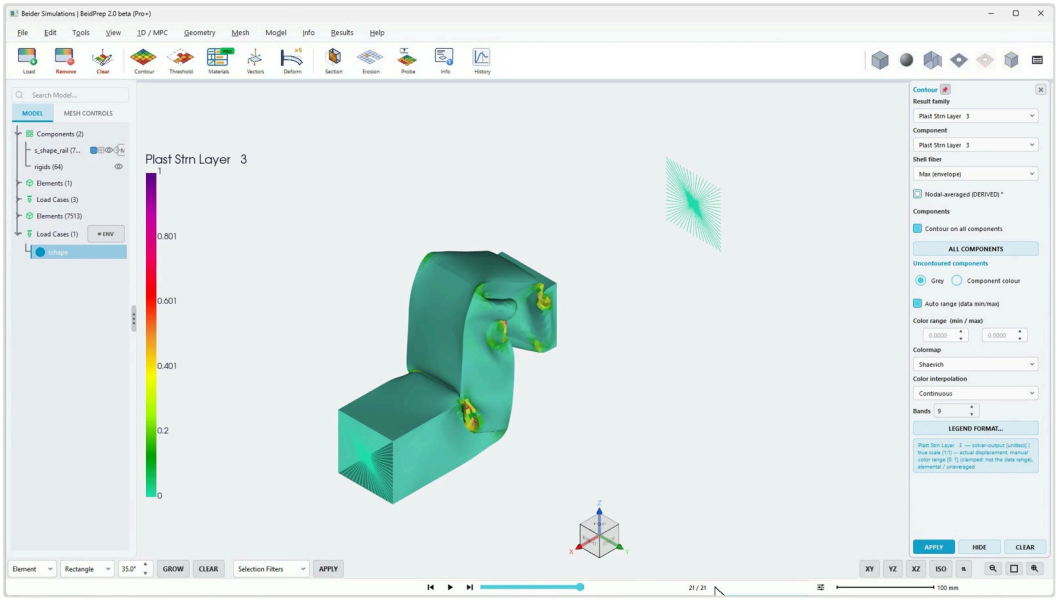
video 0:37

1. Press Play.

   The rail folds, and the colours follow each moment.

2. Drag the time slider to go to any moment.

   The readout shows which state is shown.



The screen at 0:48 in the video.

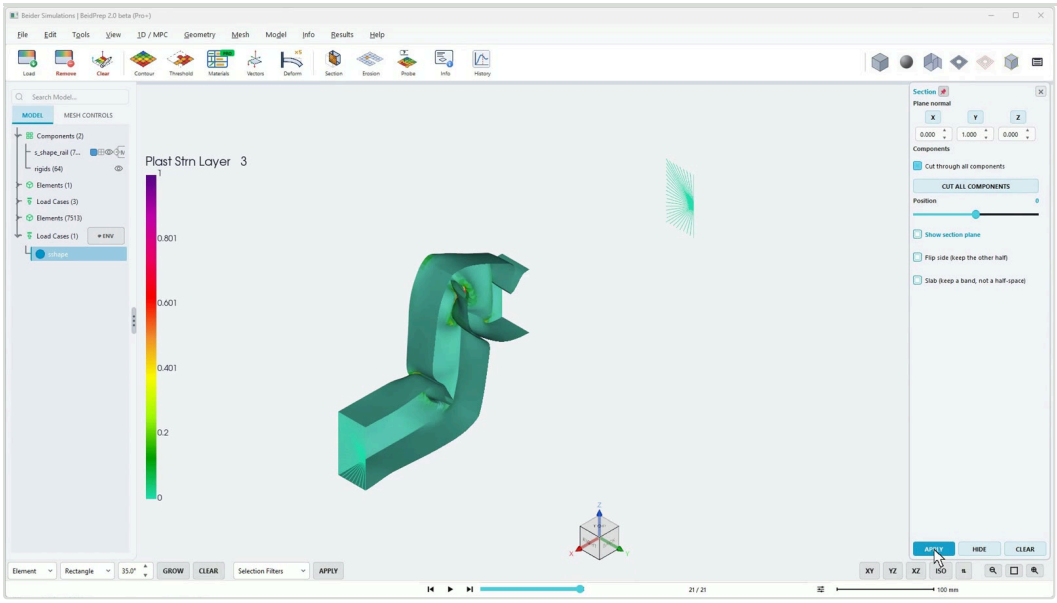
5 Deform, Section

video 0:48

Deform shows the deformed shape, and can scale it.

Section cuts the model with a plane, to see inside it.

1. Set the plane normal to Y, and press Apply.



The screen at 1:03 in the video.

6 Threshold

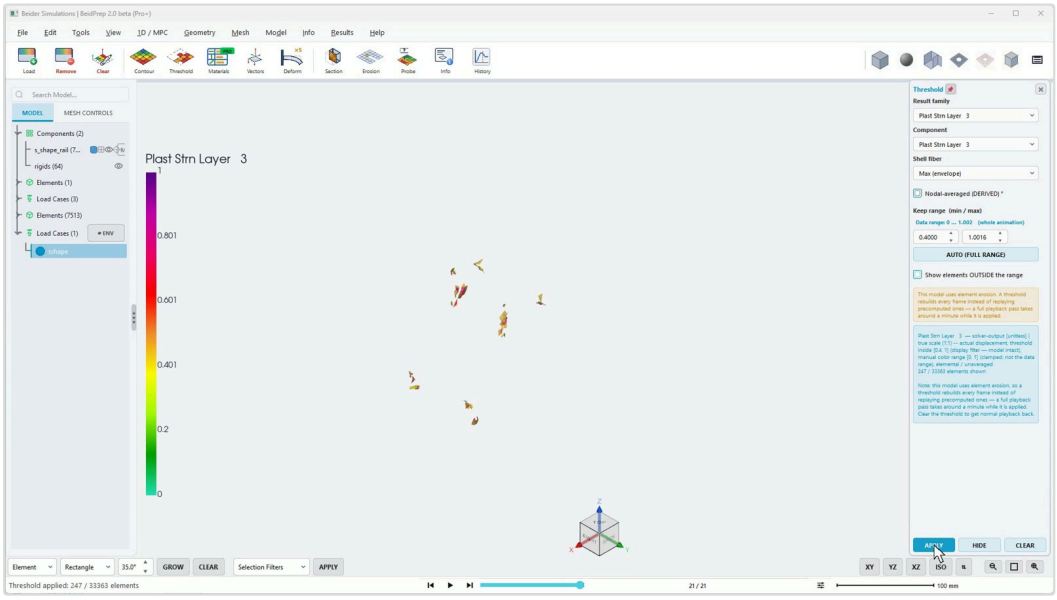
video 1:07

Threshold shows only the elements inside a range you set.

1. Choose plastic strain, and set the minimum to zero point four.
2. Press Apply: only the elements at or above zero point four stay visible.

Clear brings every element back.

In the video: **0.4** in lower value.



The screen at 1:19 in the video.

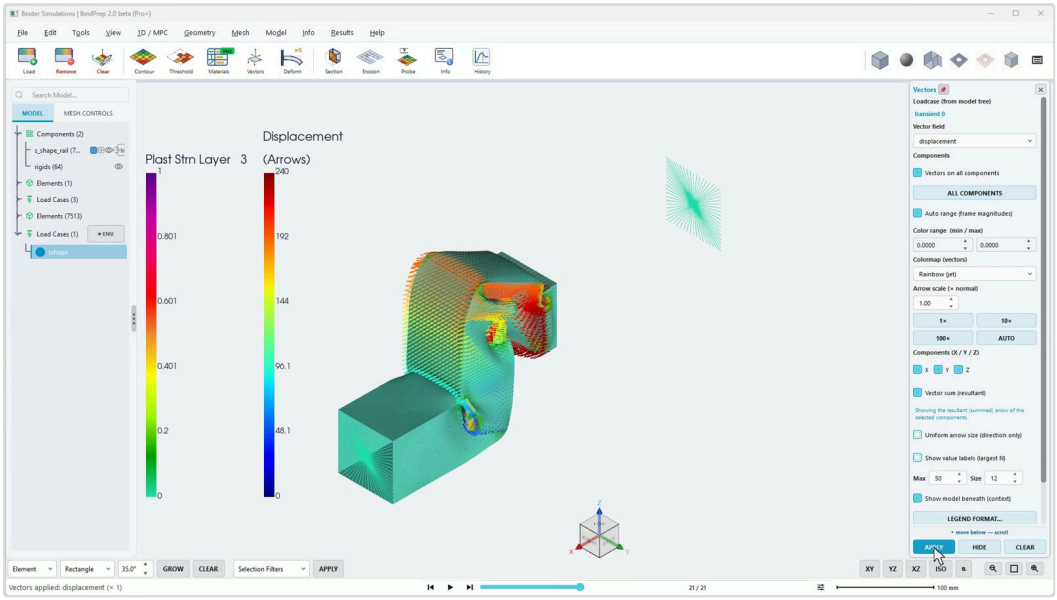
7 Vectors

video 1:22

Vectors can draw any vector result as arrows.

Here it shows displacement.

1. Press Apply, and the arrows appear.



The screen at 1:32 in the video.

8 Probe, Info

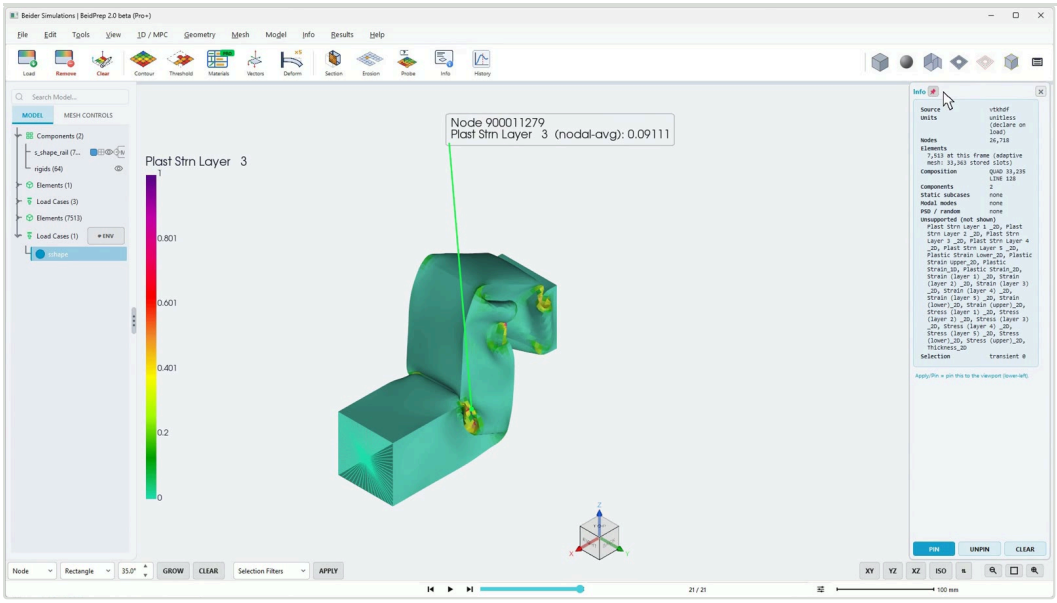
video 1:35

Probe reads the values at one point.

1. Click an element on the fold.

Info sums up the model, and the selection.

It can stay pinned on the view.



The screen at 1:48 in the video.

9 History, Clear, Close

video 1:49

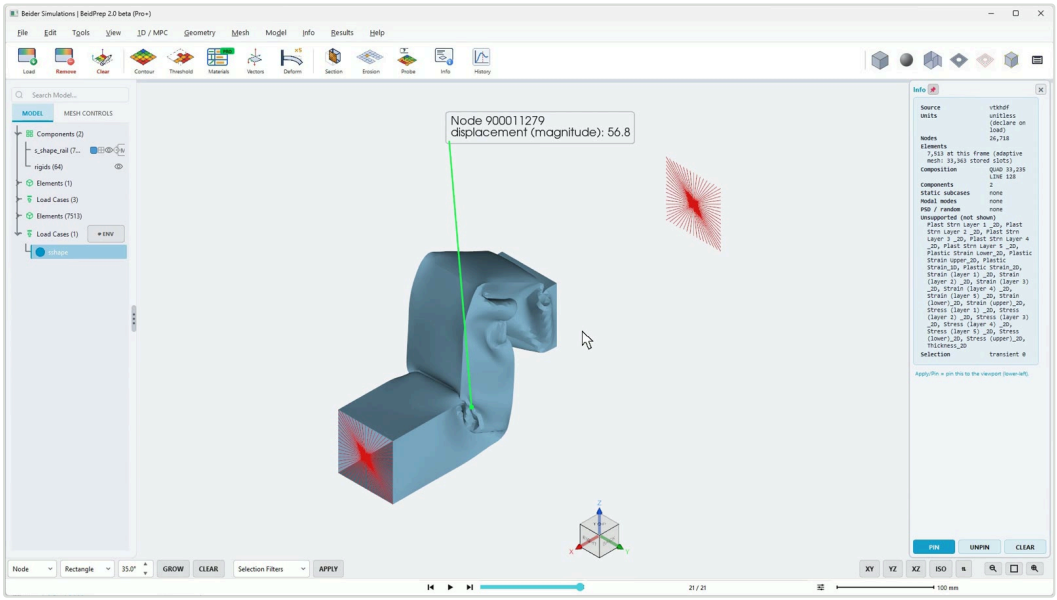
History plots the run's time history: reaction forces and energies.

Video eleven shows it in full.

Clear removes the colours and filters.

The model and the deformed shape stay.

That is the Results tab: colours, shape, cuts and values, through the whole run.



The screen at 2:07 in the video.

Steps and pictures are taken from tutorial video 10. Every picture is a frame of the video.

TUTORIAL VIDEO 11

11 Time histories: energies, contact force, peaks

Plot time histories from an OpenRadioss T01 file: energies, a contact force and its peak, the derivative and your own curves, two cursors, statistics, and a PNG for your report.

Tutorial files	folder `11_Time_histories` — open `wingbay_i_bird_4ms_0000.rad` first (this video also needs the results zip)
Video	Watch video 11 on YouTube
Download files	Download the tutorial files (97 MB) Download the solver results (225 MB)

1 Opening

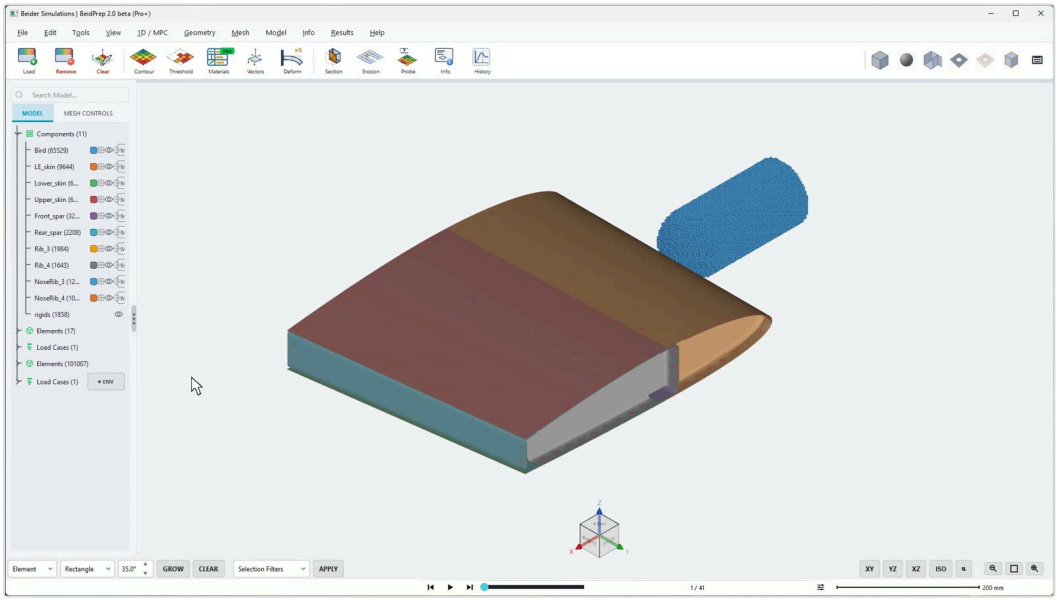
video 0:00

A bird strike run writes two kinds of results.

Animation files show the model at a few moments.

The time history records chosen values every microsecond.

This video shows how to plot it.



The screen at 0:12 in the video.

2 T01 file, History

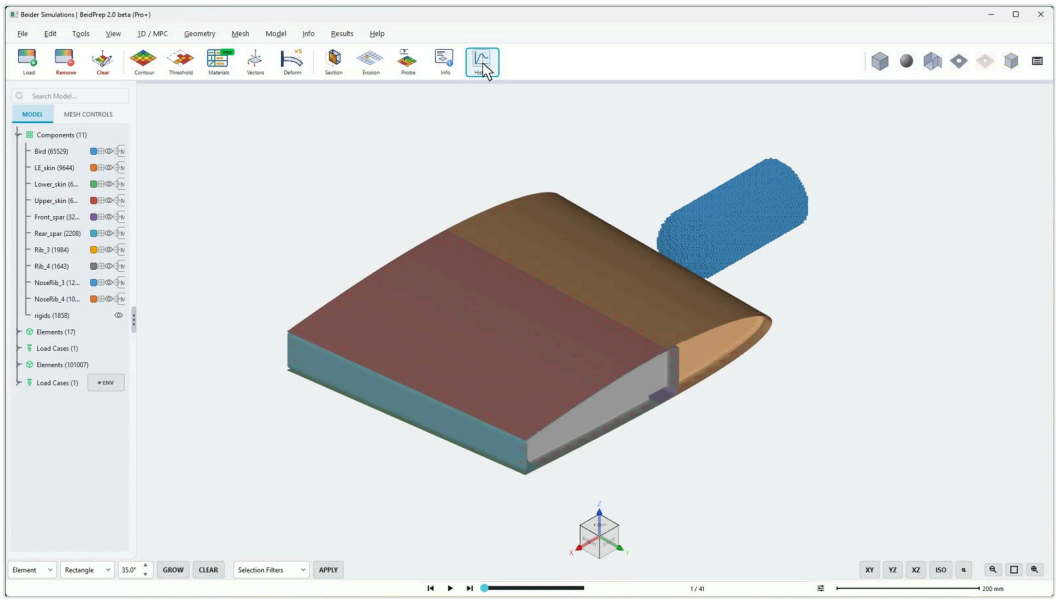
video 0:12

1. Open the run's results first, as video ten shows.

   The time history file ends in T zero one.

It sits beside the animation files.

2 On the Results tab, press History.



The screen at 0:24 in the video.

3 Run line, Curves

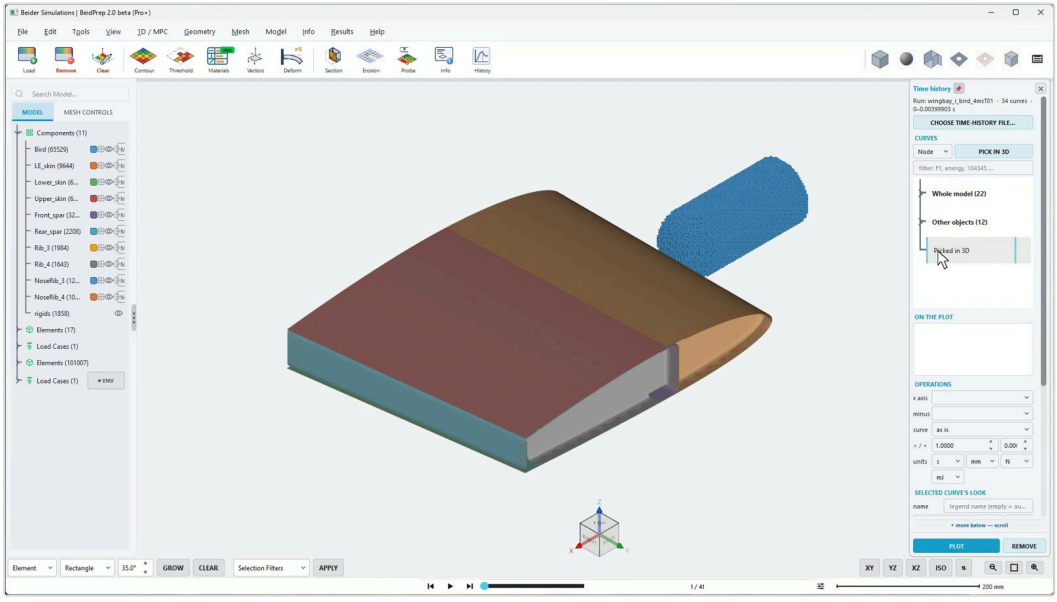
video 0:25

The panel opens on the right, and reads the file at once.

Run, at the top, names the file it read.

Under Curves, every recorded curve is listed by object.

Each group shows how many curves it holds.



The screen at 0:38 in the video.

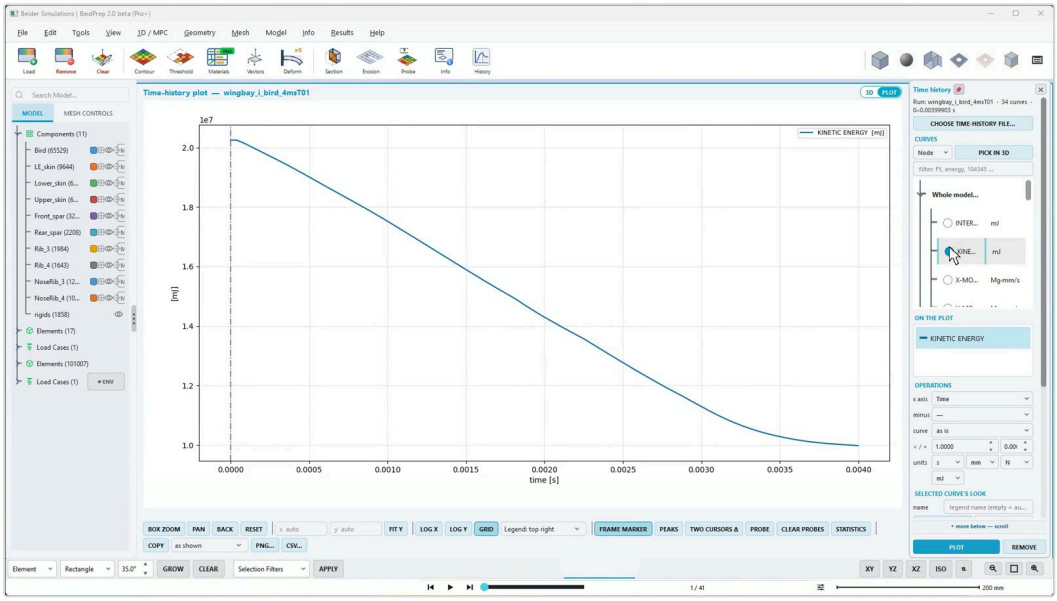
4 Whole model, Kinetic

video 0:38

Whole model holds the energies of the run.

1. Open it.
2. Tick kinetic energy.

   The plot replaces the 3D view at once.



The screen at 0:48 in the video.

5 Internal

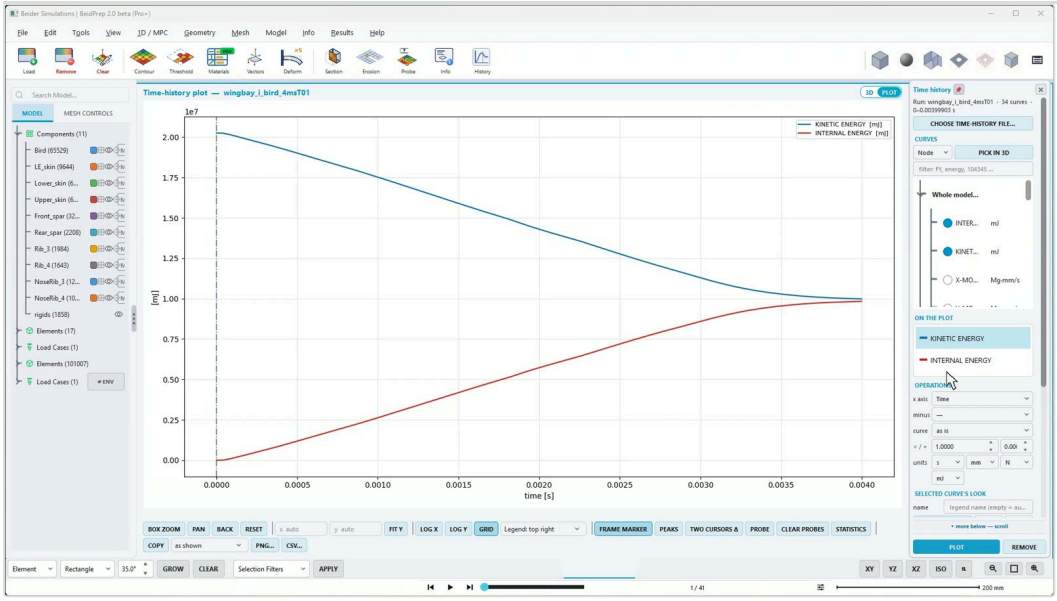
video 0:48

1. Tick internal energy as well.

   The bird slows down.

   Kinetic energy falls, and internal energy rises.

   It goes into the skin and the spar.



The screen at 0:59 in the video.

6 Hourglass, Click to jump

video 0:59

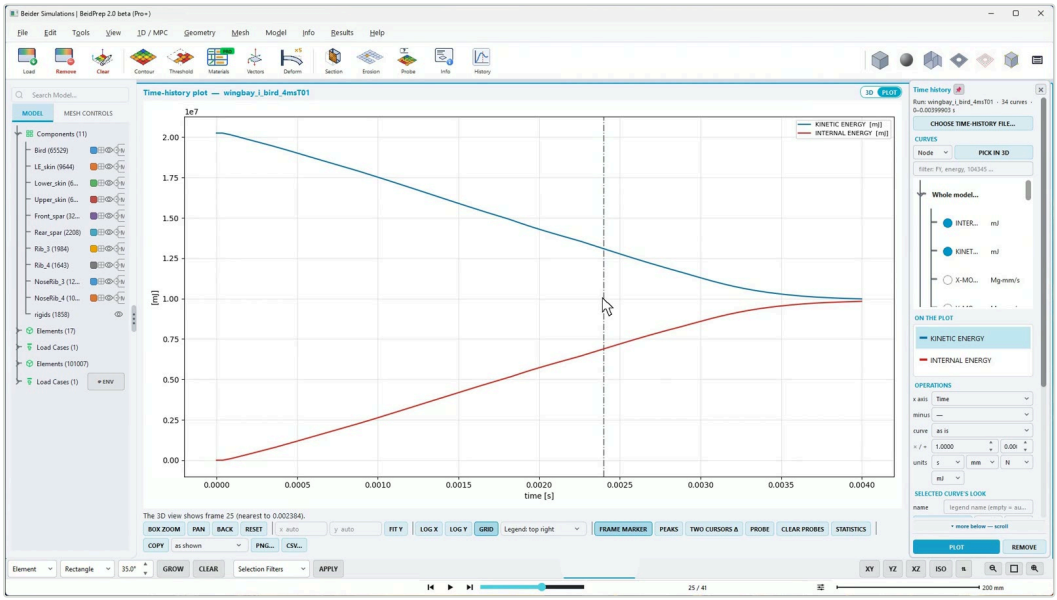
1. Tick hourglass energy.

   It stays under two percent of the start.

   That is what you want in a shell model.

   A grey dash-dot line marks the frame shown in 3D.

2. Click on the plot, and the 3D view jumps to that time.



The screen at 1:13 in the video.

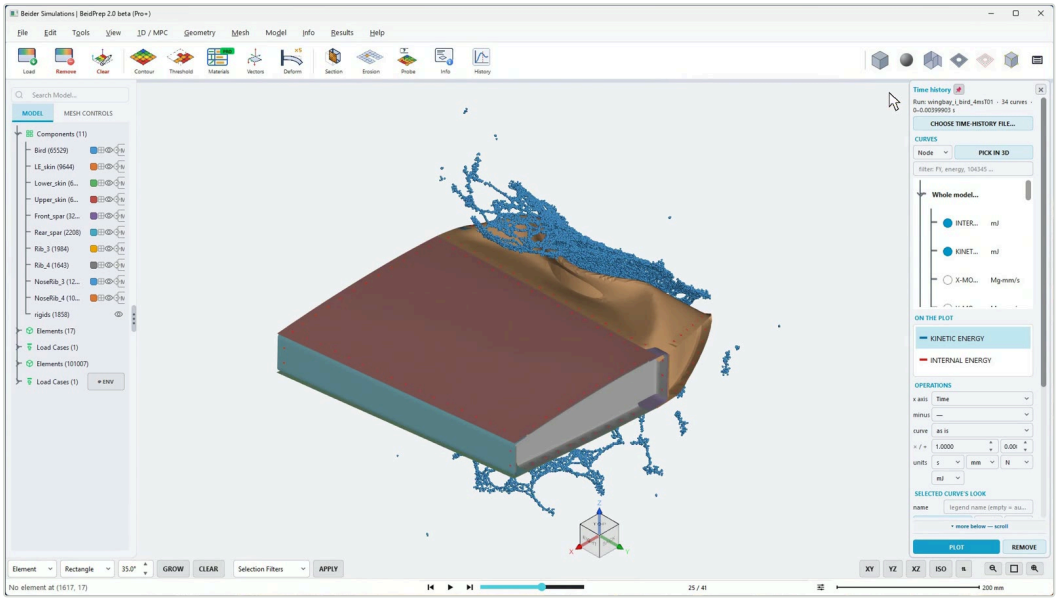
7 Back to 3D

video 1:13

The top right switch goes back to the 3D view.

Here is the wing at that moment, just as the skin starts to tear.

1. Switch back to the plot.



The screen at 1:22 in the video.

8 Contact force, Impulse

video 1:23

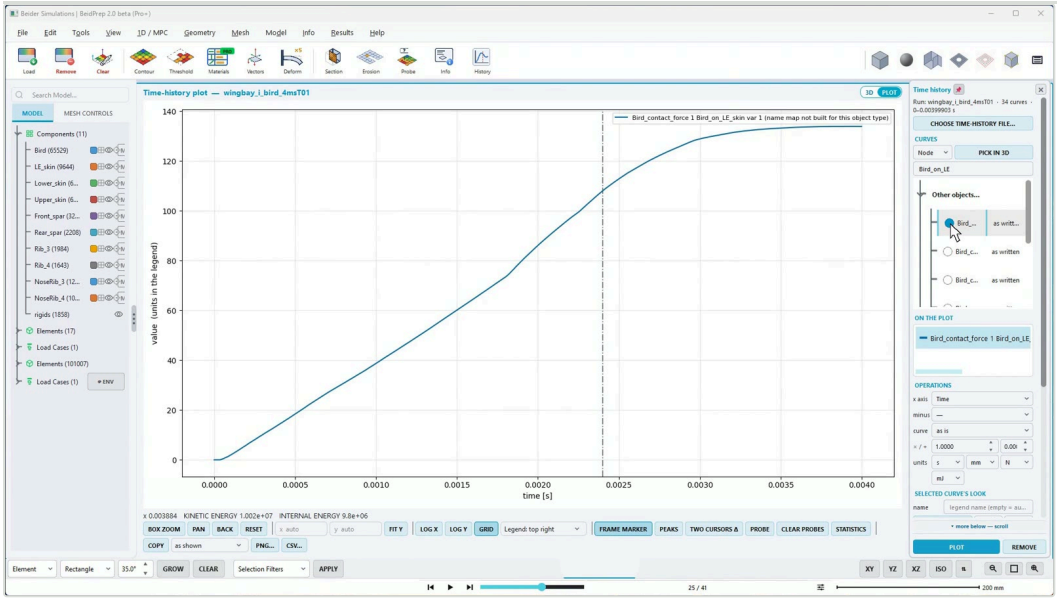
Now the force of the bird on the skin.

The contact curves are under Other objects.

The solver writes this force as an impulse, so the curve only grows.

Var one is the normal force along x, the flight direction.

In the video: **Bird_on_LE** in filter box.



The screen at 1:38 in the video.

9 d/dt, Scale and name

video 1:38

1. Under Operations, choose d by d t.

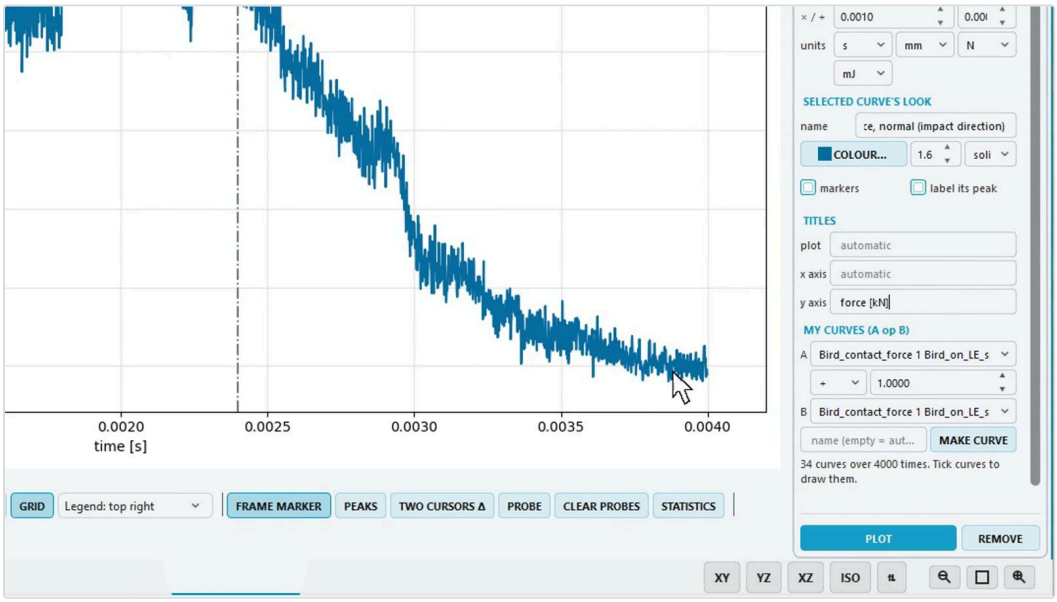
   The curve becomes the force.

2. Scale it by one thousandth to read kilonewtons, and give the curve a name.

   In the video: **0.001** in x / + scale field.

   In the video: **Contact force, normal (impact direction)** in Selected curve's look -> name.

   In the video: **force [kN]** in Titles: y-axis title.



The screen at 1:49 in the video.

10 Peaks, Probe

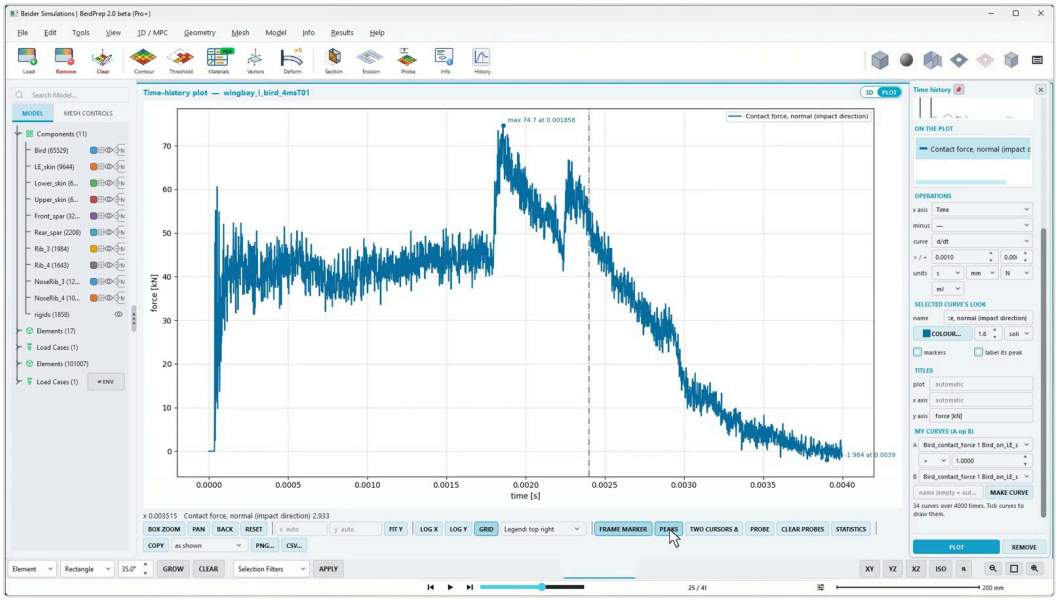
video 1:49

1. Press Peaks.

   The largest force is about seventy-five kilonewtons.

   It comes at one point nine milliseconds, just before the skin tears.

   Probe reads a value only on the curve: a click beside it finds no curve.



The screen at 1:59 in the video.

11 Two cursors, Box zoom

video 2:04

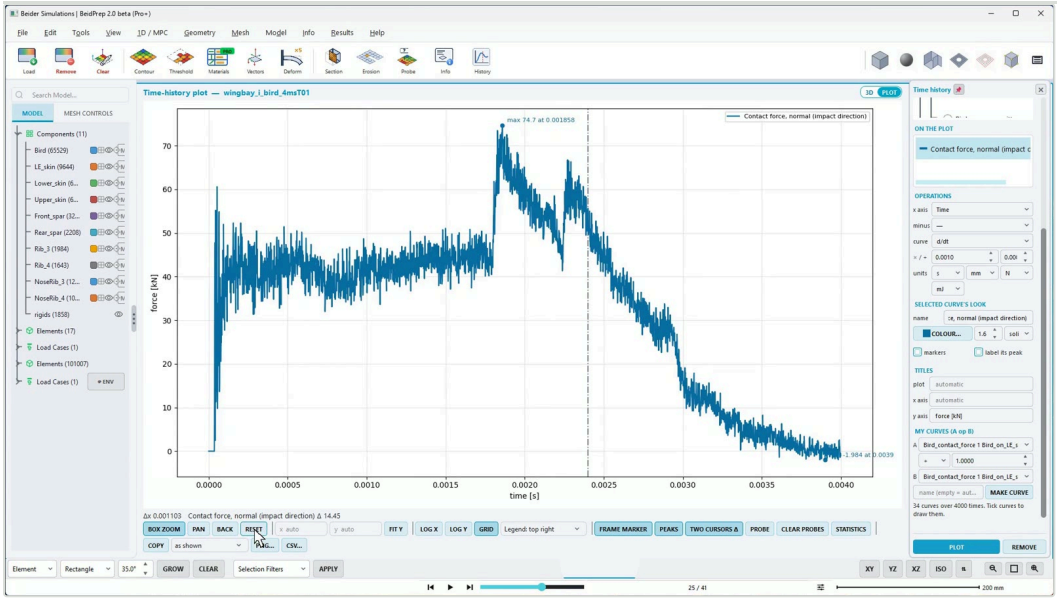
1. To compare two moments, press Two cursors and click twice.

   The differences appear under the plot.

   Box zoom looks closer.

2. Drag a box around the peak.

   Reset brings the full view back.



The screen at 2:20 in the video.

12 Statistics, My curves

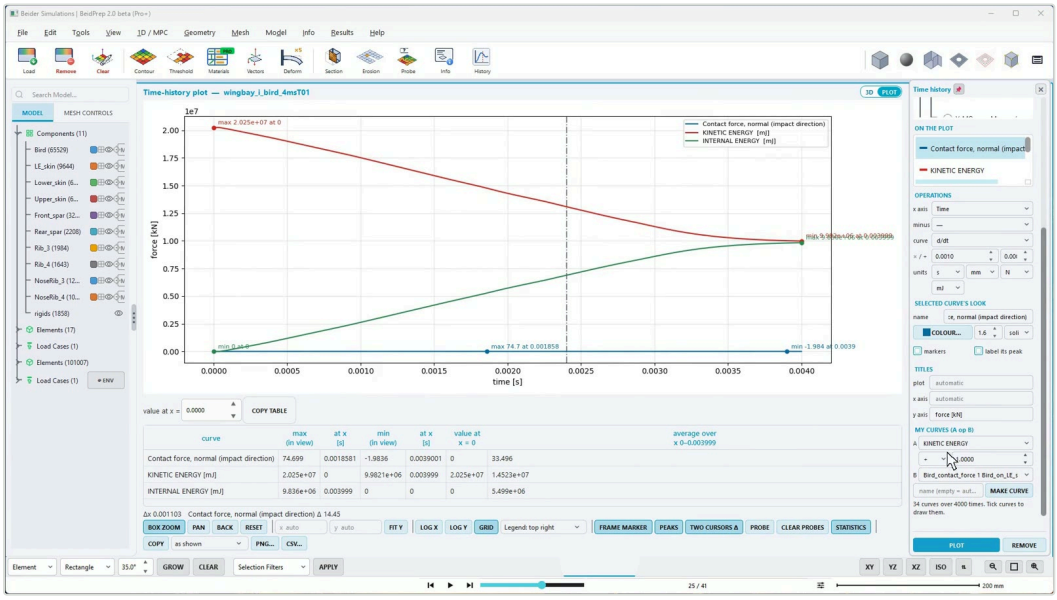
video 2:21

Statistics gives each curve's maximum, minimum and average.

It uses the range on screen.

Under My curves you can combine two curves.

1. Add kinetic and internal energy.



The screen at 2:33 in the video.

13 Energy balance, Save PNG

video 2:36

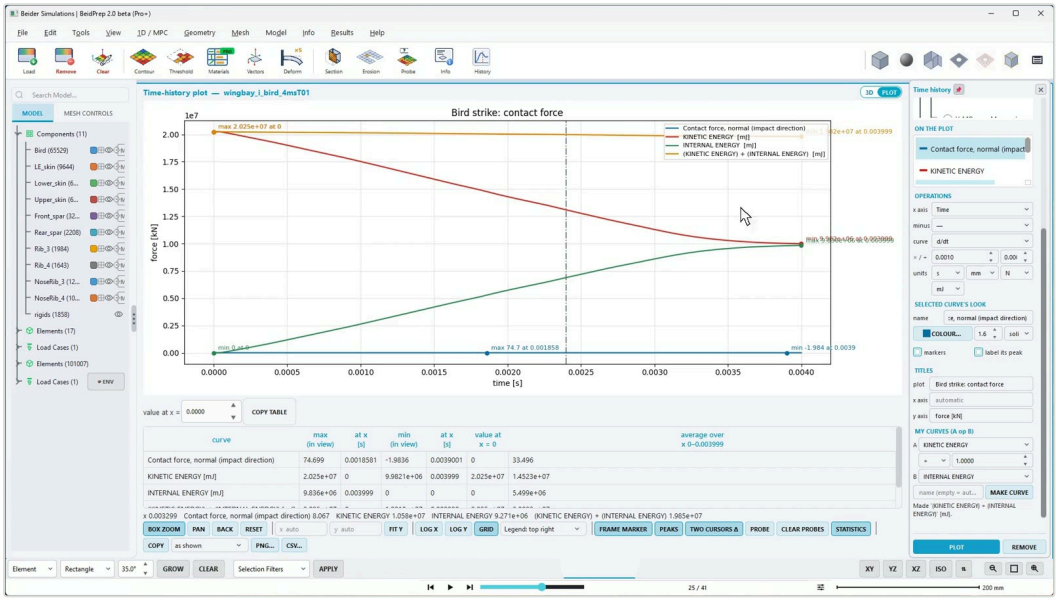
The sum stays close to the start value.

That is the energy balance check: here the error is about two percent.

1 Give the plot a title, then save it.

P N G saves a picture; C S V saves exactly the drawn curves as numbers.

In the video: **Bird strike: contact force** in Titles: plot title.



The screen at 2:54 in the video.

14 Close, Summary

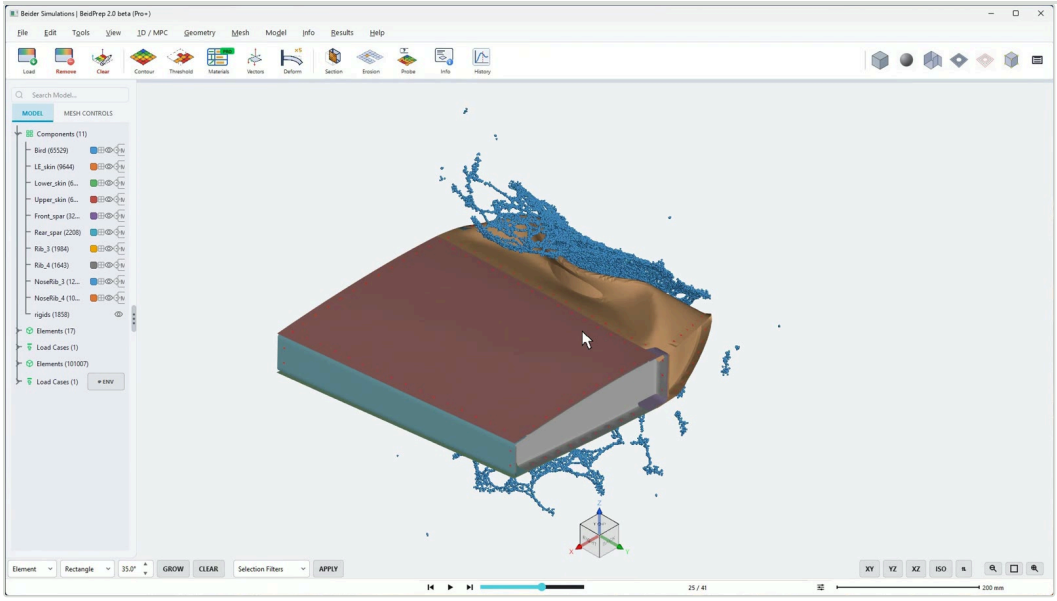
video 2:54

1 Press the cross at the top to close the tool.

The 3D view comes back.

That is the History tool: curves by object, operations, probes and statistics.

It all comes from the file the solver wrote.



The screen at 3:08 in the video.

Steps and pictures are taken from tutorial video 11. Every picture is a frame of the video.

TUTORIAL VIDEO 12

12 The fibre module: short-fibre composites from tensile curves

Calibrate a short-fibre composite from its tensile curves, map Moldflow fibre orientation onto the part, export the mapped materials and check them against a tensile run.

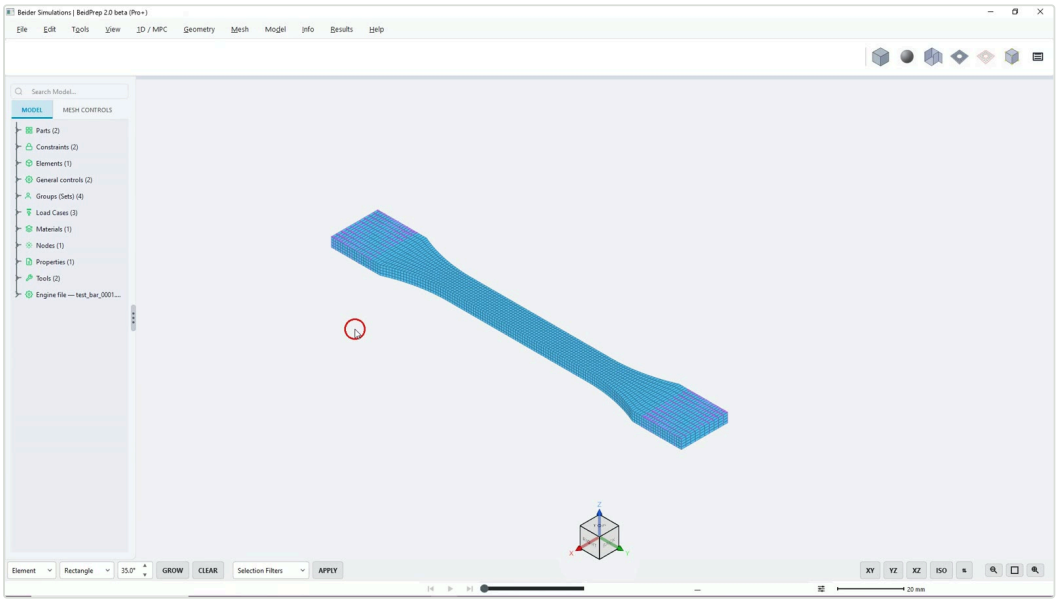
Tutorial files	folder `12_Fibre_module` — open `deck\test_bar_0000.rad` first (this video also needs the results zip)
Video	Watch video 12 on YouTube
Download files	Download the tutorial files (97 MB) Download the solver results (225 MB)

1 Opening

video 0:00

This video brings the fibre directions from the Moldflow filling results onto a test bar.

The material is PA6 GF50, 50 % glass fibre, calibrated here from its own tensile curves.



The screen at 0:14 in the video.

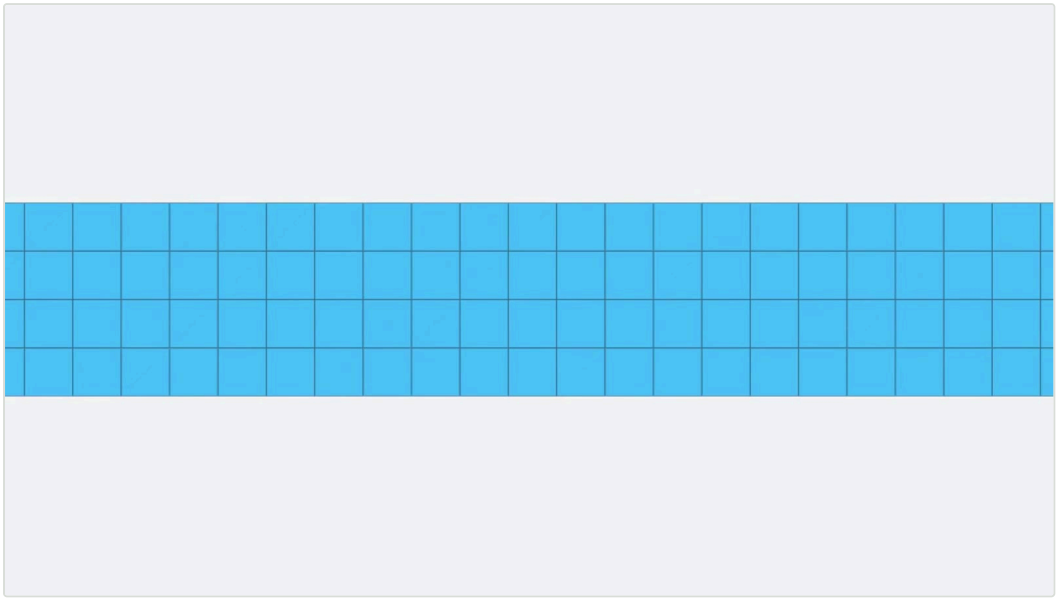
2 The case, Hexa mesh

video 0:14

The bar is cut from a plate that was filled from two gates.

Where the two flow fronts meet they leave a weld line, and here it crosses the bar's narrow section.

The bar comes in already meshed with hexahedra, four layers through its four millimetres: the fibre module works on solid elements, and BeidPrep 2.0 doesn't generate solid meshes yet.



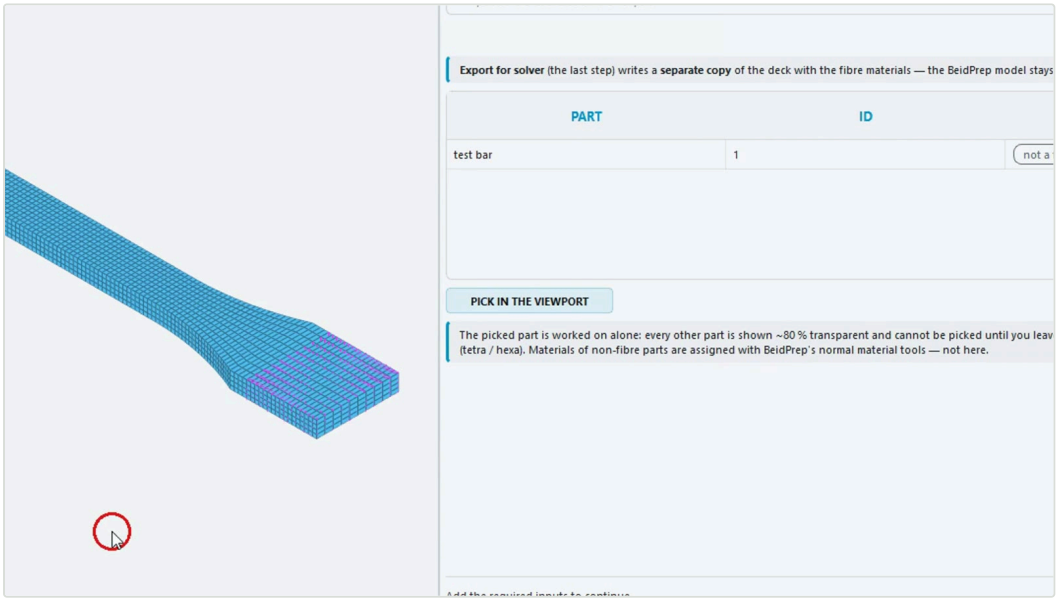
The screen at 0:36 in the video.

3 Open the module

video 0:37

1. On the Model tab, click Fibre module.

   The panel works in four steps: the part, the material, the orientation, and the export.



The screen at 0:47 in the video.

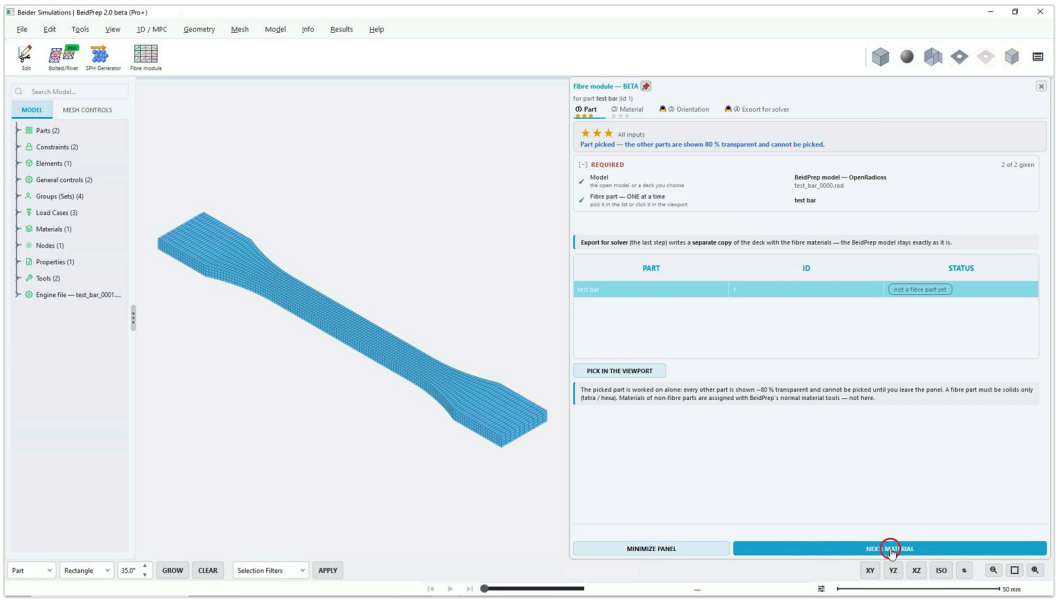
4 Part

video 0:47

Step one is the part.

1. Click the bar: the module works on one short-fibre part at a time, meshed with solids.

   Your model is not changed. The last step writes a copy for the solver.



The screen at 0:59 in the video.

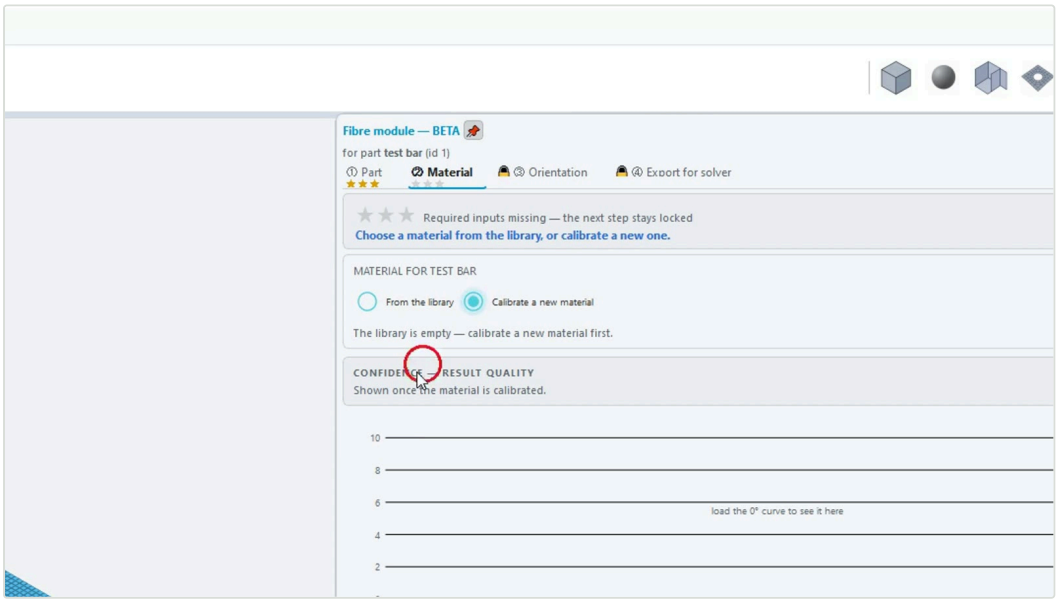
5 Material

video 1:00

Step two is the material.

1. Pick one already calibrated from the library, or calibrate a new one from your own tensile curves.

   This video starts from the curves.



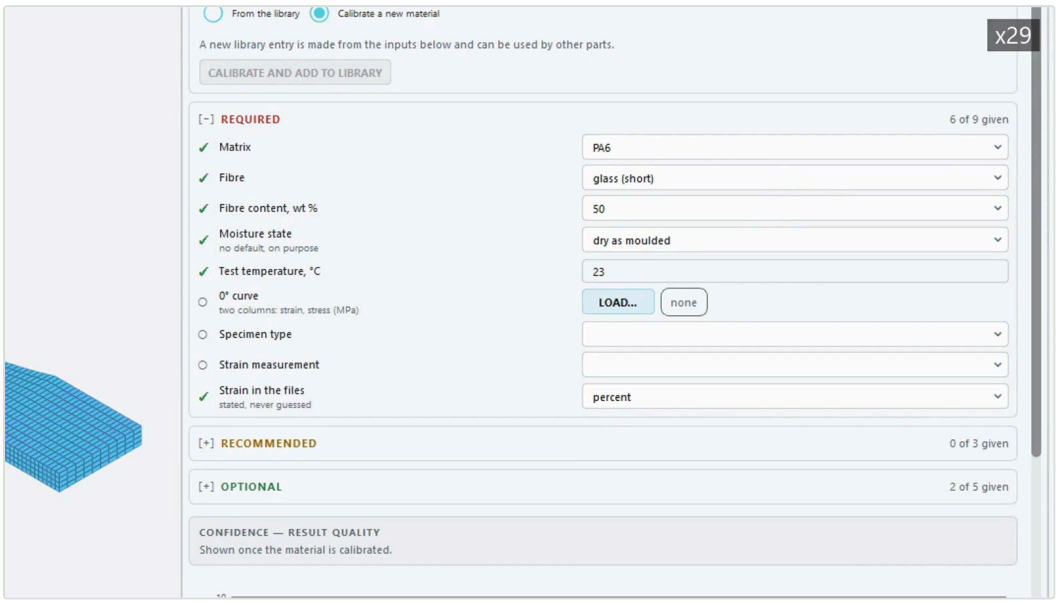
The screen at 1:11 in the video.

6 Required

video 1:13

The required inputs describe the material and the test.

Polyamide 6, short glass fibre, fifty percent by weight, dry as moulded, tested at twenty three degrees.



The screen at 1:24 in the video.

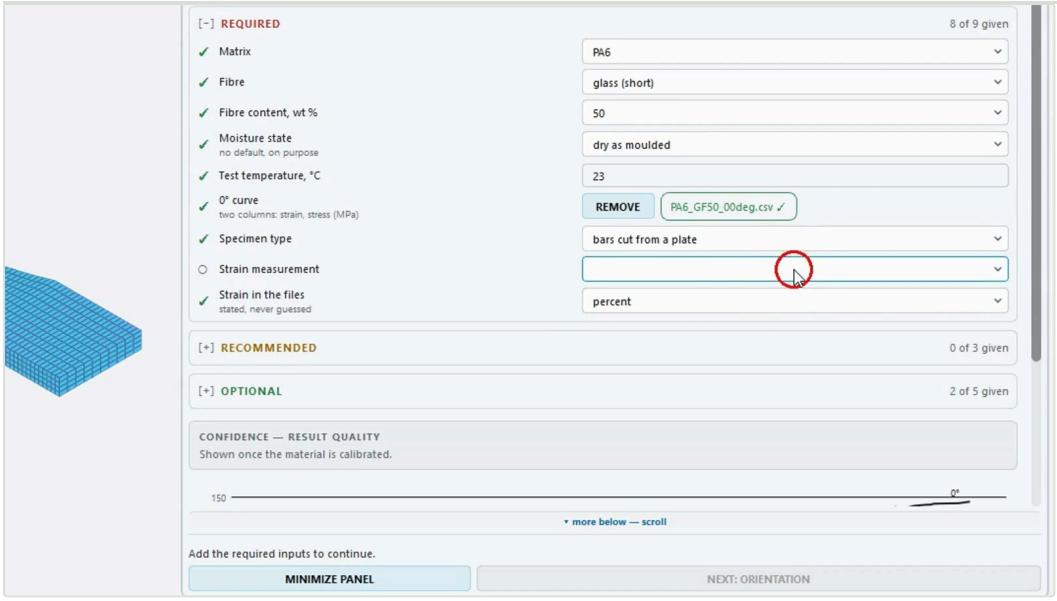
7 0° curve

video 1:25

1. Load the zero degree curve, pulled along the flow.

   The bars were cut from a plate, the strain was measured with an extensometer, and the file gives it in percent.

   In the video: **PA6_GF50_00deg.csv** in file dialog.



The screen at 1:36 in the video.

8 Recommended, Confidence LOW

video 1:37

The forty five and ninety degree curves are recommended, not required.

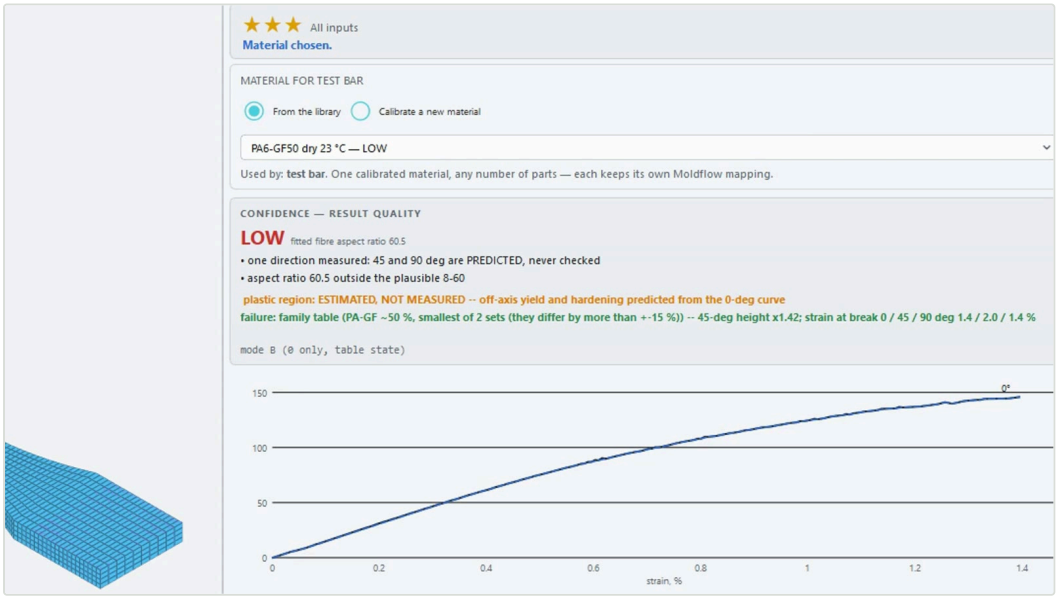
1. Leave them empty for now, and press Calibrate and add to library.

The confidence card says LOW, and it says why.

Only one direction was measured: forty five and ninety degrees are predicted, never checked.

And the fitted fibre aspect ratio, sixty point five, is outside the plausible range of eight to sixty.

The plastic behaviour is estimated, not measured.



The screen at 2:06 in the video.

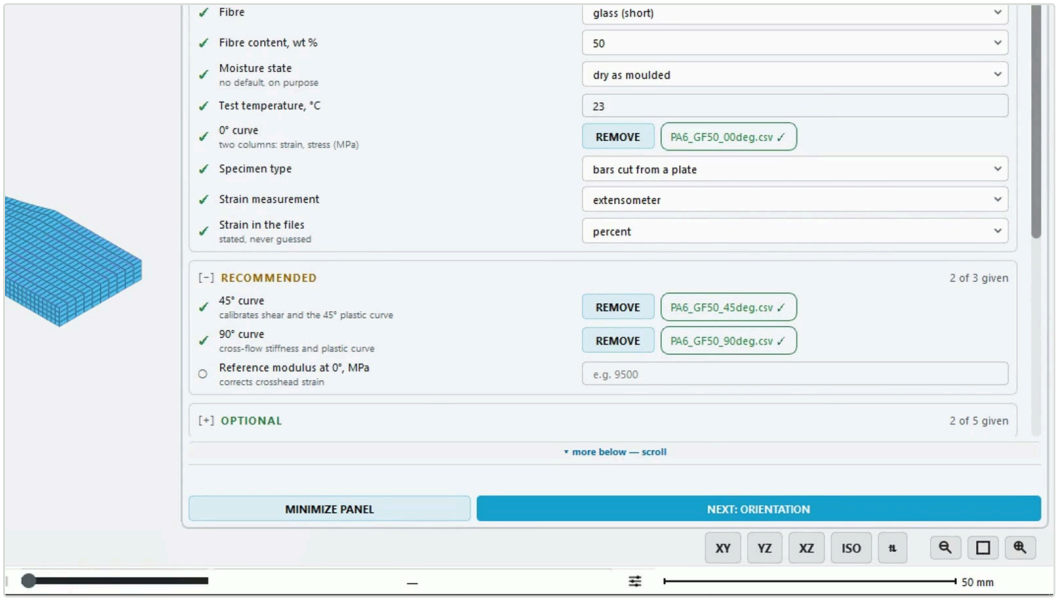
9 Add 45° and 90°

video 2:06

1. Now add the forty five and ninety degree curves.
2. Choose Calibrate a new material again: the form keeps what you entered.
3. Load both curves, and calibrate again.

   In the video: **PA6_GF50_45deg.csv** in file dialog.

   In the video: **PA6_GF50_90deg.csv** in file dialog.



The screen at 2:20 in the video.

10 Confidence MEDIUM

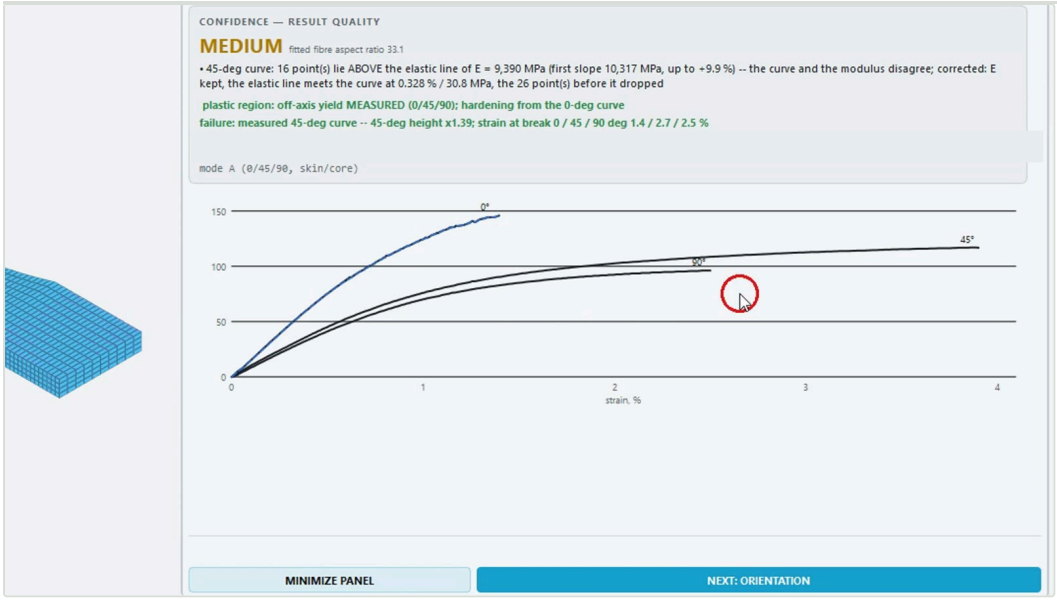
video 2:24

The confidence rises to MEDIUM.

The fitted aspect ratio is now thirty three, inside the plausible range, and the yield off the fibre axis is measured.

One reason is left: the forty five degree curve starts above its elastic line.

The plot shows the three measured curves, and in blue, dashed, the master curve the solver uses.



The screen at 2:46 in the video.

11 Library, Moldflow files

video 2:46

Both calibrations are now in the library, under the same name, each with its confidence.

1. Keep the MEDIUM one for this bar.

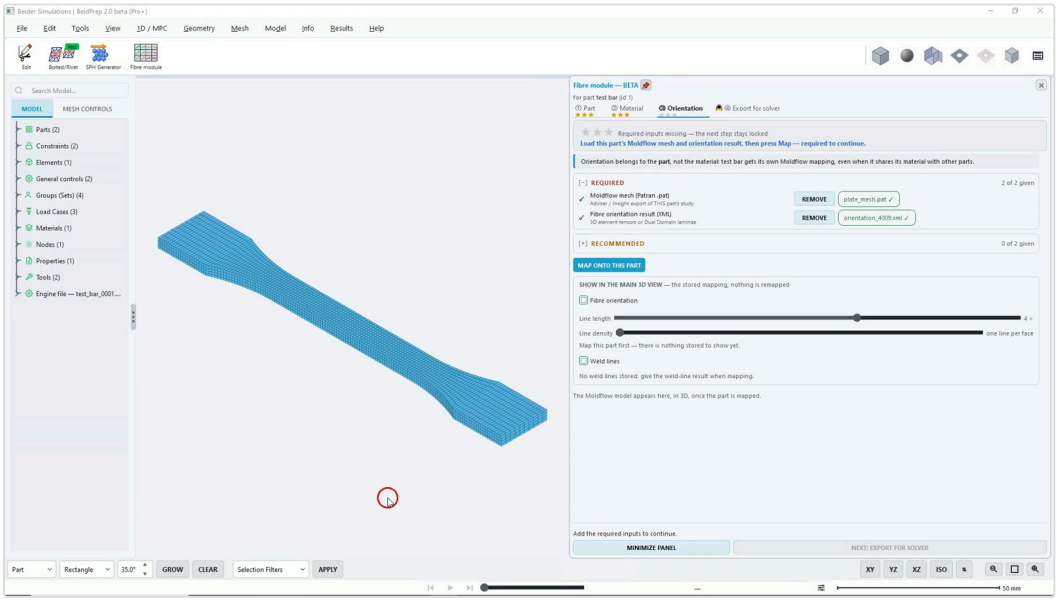
   Step three is the orientation.

2. Load the Moldflow mesh, and the fibre orientation result.

   The orientation belongs to the part: another part can use the same material with its own mapping.

   In the video: **plate_mesh.pat** in file dialog.

   In the video: **orientation_4009.xml** in file dialog.



The screen at 3:11 in the video.

12 Weld-line file, Map

video 3:11

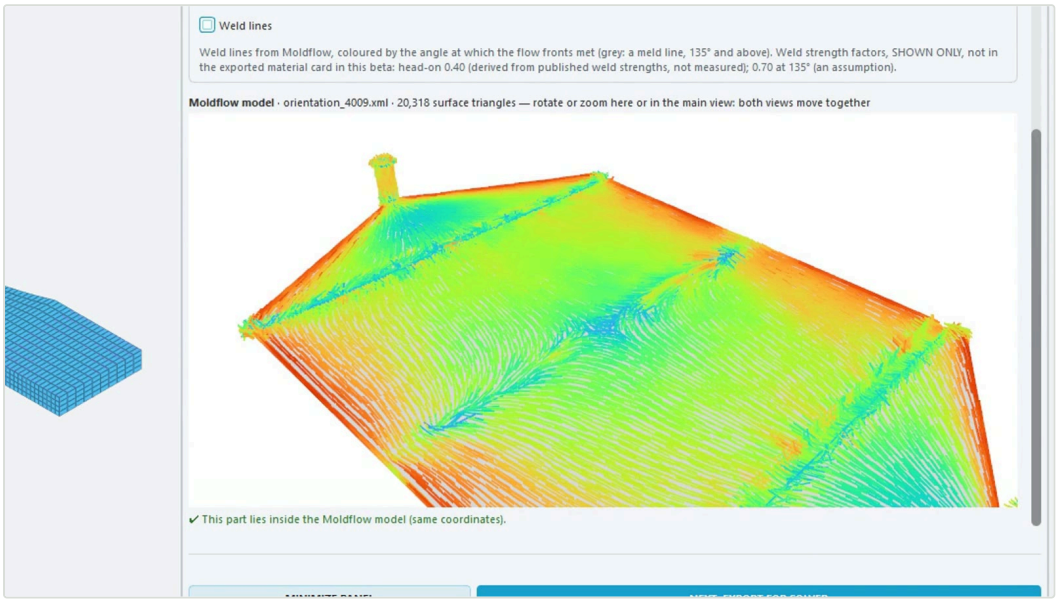
The weld-line result is optional.

1. Load it in its own row.
2. Press Map onto this part.

BeidPrep first checks that the bar lies inside the plate, then gives every element its fibre direction.

The linked view now shows the plate the results come from.

In the video: **weld_lines_1722.nod** in file dialog.



The screen at 3:35 in the video.

13 Fibre colours, Weld lines

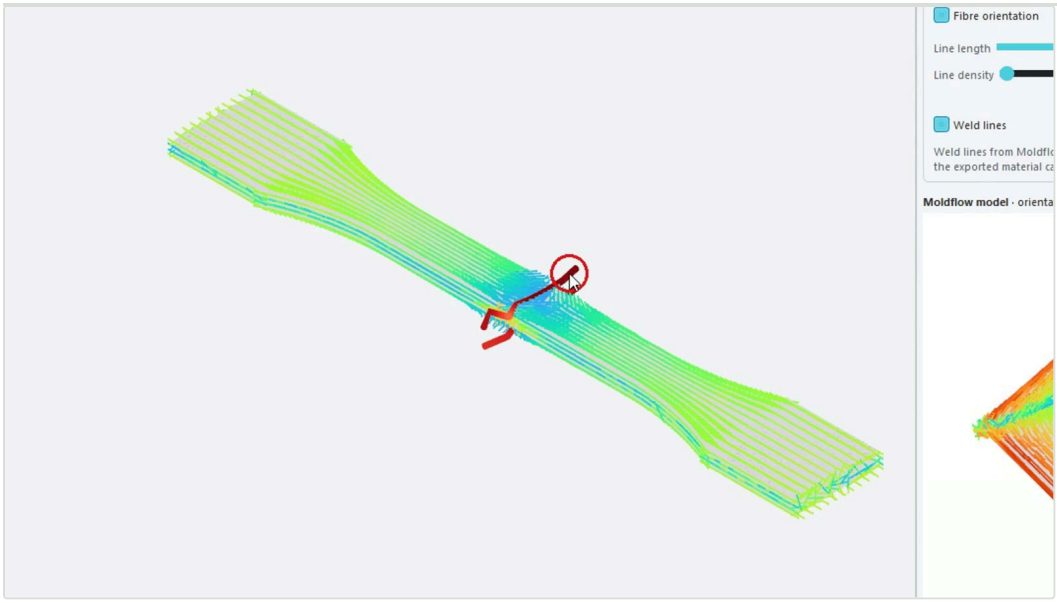
video 3:36

Under Show in the main 3D view, Fibre orientation colours the bar by fibre direction.

Tick Weld lines.

The weld line is drawn as a tube across the narrow section, coloured by the angle at which the flow fronts met.

The legend gives the angles.



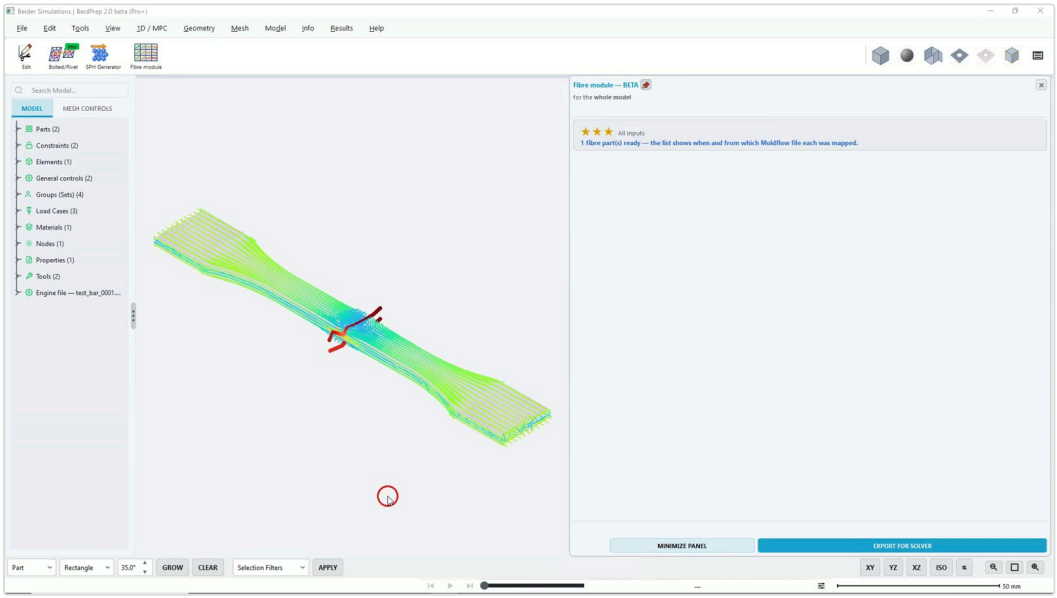
The screen at 3:51 in the video.

14 Export

video 3:54

Step four is the export.

Export for solver writes a copy of the deck for OpenRadioss: the fibre elements are sorted into material groups, and each element keeps its own fibre direction.



The screen at 4:07 in the video.

15 Tensile run, Strain

video 4:08

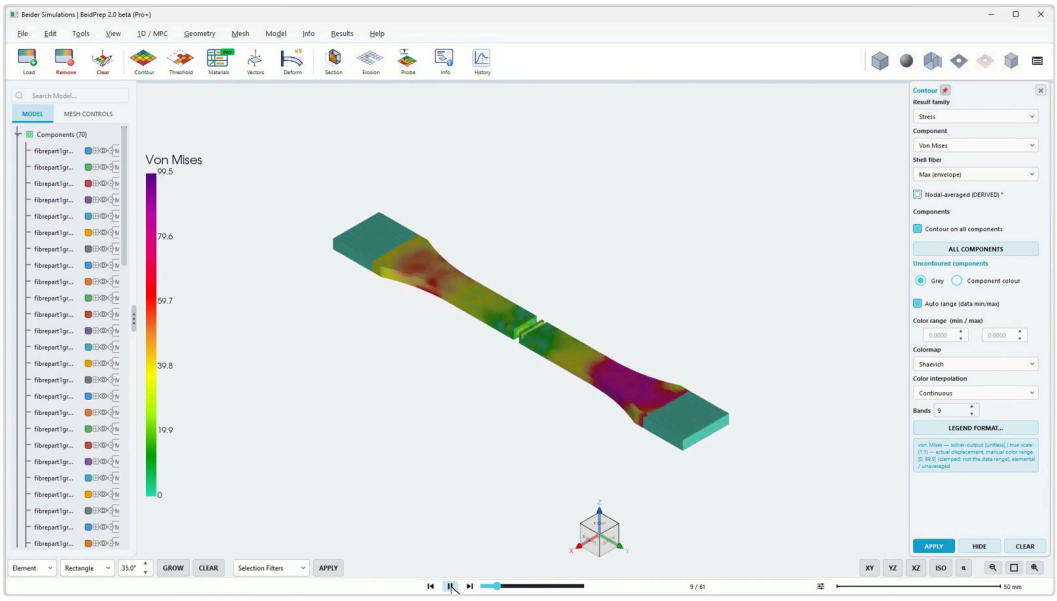
The bar is then pulled in OpenRadioss, like a tensile test.

1. Click Load on the Results tab, and open the run.

Contour shows the von Mises stress.

2 Play the run, and watch where the strain gathers.

In the video: **test_bar_fibreA001** in file dialog.



The screen at 4:25 in the video.

16 Force against time

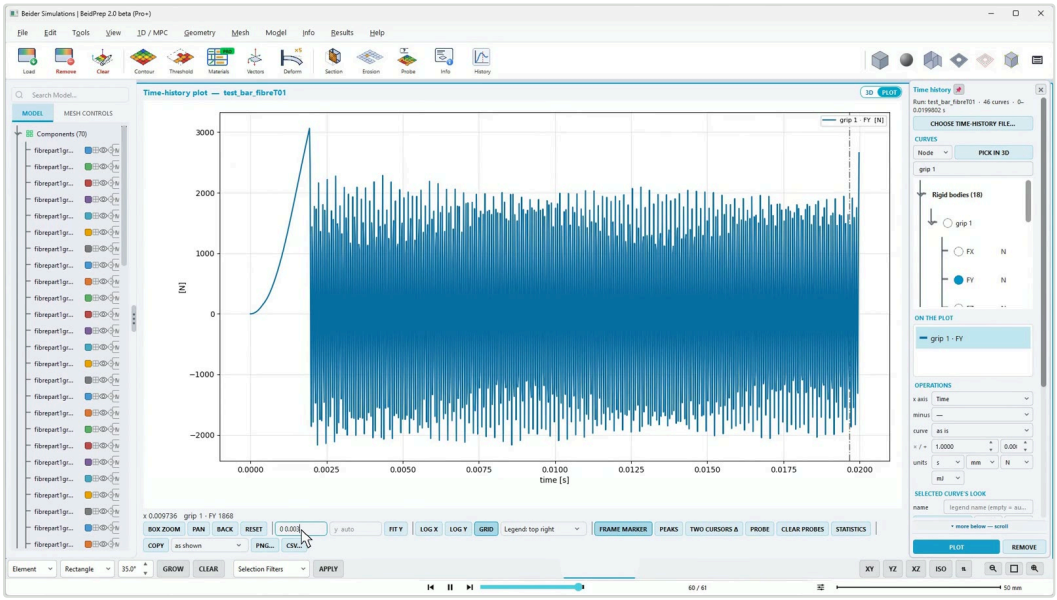
video 4:27

History plots the grip force against time.

1 Show the first three milliseconds, where the bar is pulled and breaks.

In the video: **grip 1** in filter box.

In the video: **0 0.003** in x range field.



The screen at 4:42 in the video.

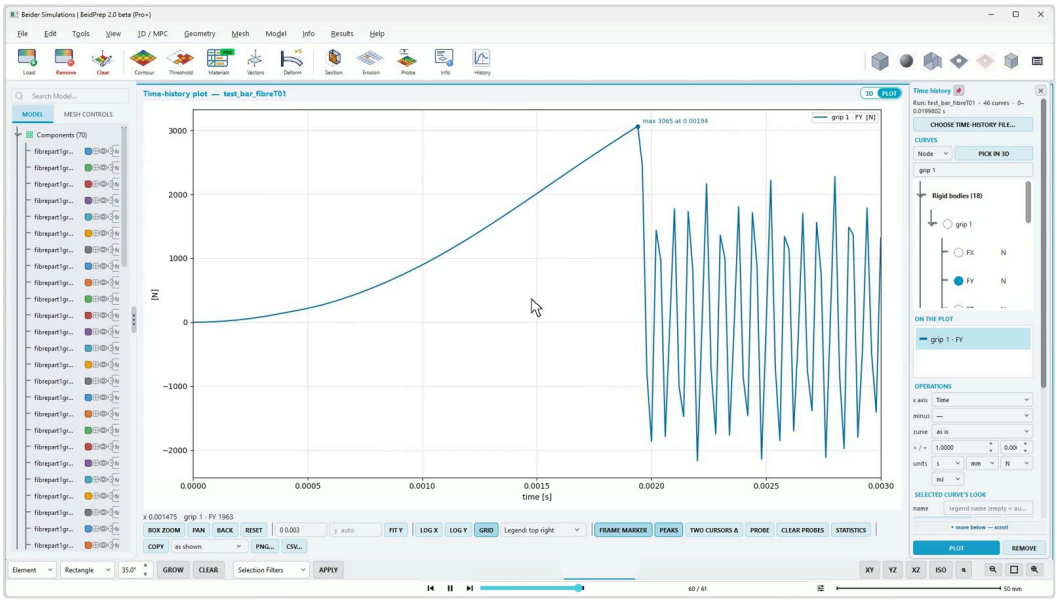
17 Peak, Close

video 4:44

Peaks marks the highest force: about three thousand and sixty newtons, at just under two milliseconds.

Then the bar breaks: the force drops, and the grip only rings around zero.

That is the fibre module: a material calibrated from your own curves, and the Moldflow filling results mapped onto your part, with its weld lines.



The screen at 5:05 in the video.

Steps and pictures are taken from tutorial video 12. Every picture is a frame of the video.

TUTORIAL VIDEO 13

13 SPH Generator: a bird strike in four steps

Set up a bird strike in four steps with the SPH Generator: the bird's size, its speed, what it hits and where it aims -- BeidPrep creates the bird, its material, the contact and the force output.

Tutorial files	folder `13_SPH_bird_strike` — open `beider_wing_bay_0000.rad` first
Video	Watch video 13 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, Open

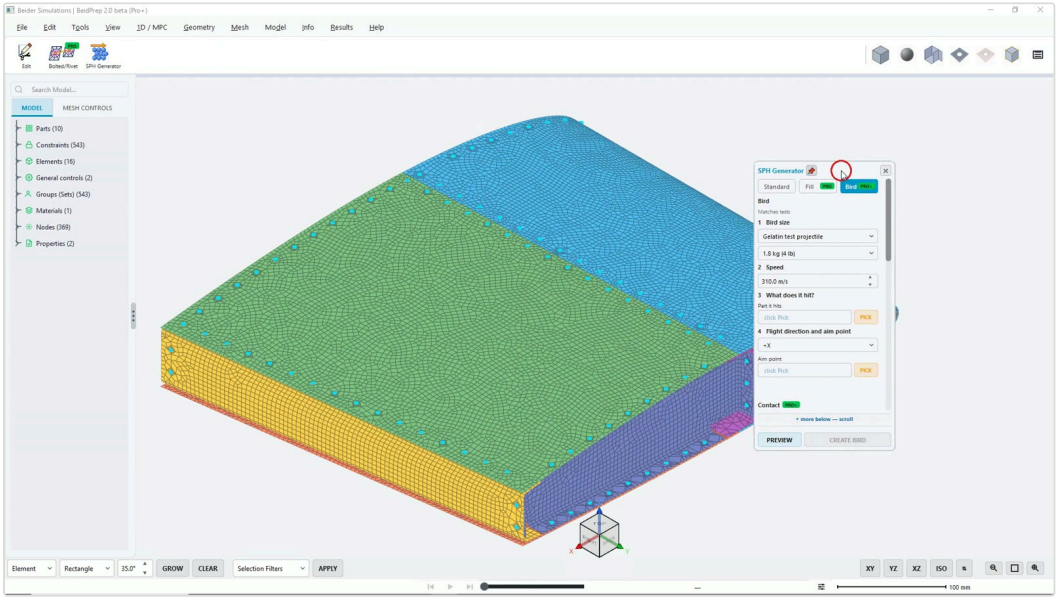
video 0:00

This video sets up a bird strike on a wing with the SPH Generator.

It takes four steps.

1. On the Model tab, click SPH Generator.

The panel opens over the 3D view. Drag it aside by its title.



The screen at 0:15 in the video.

2 Bird tab, Size

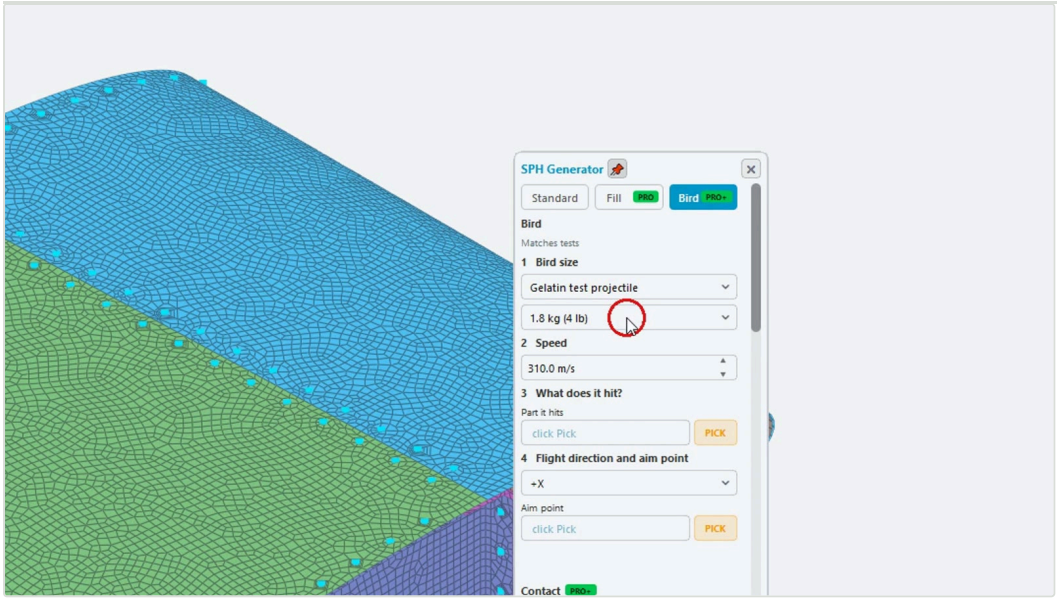
video 0:15

The Bird tab is already open.

It creates the bird, its material, its speed, the contact and the force output.

Step one is the bird's size.

1. Keep one point eight kilograms, the bird that matches tests.



The screen at 0:28 in the video.

3 Speed, Target

video 0:29

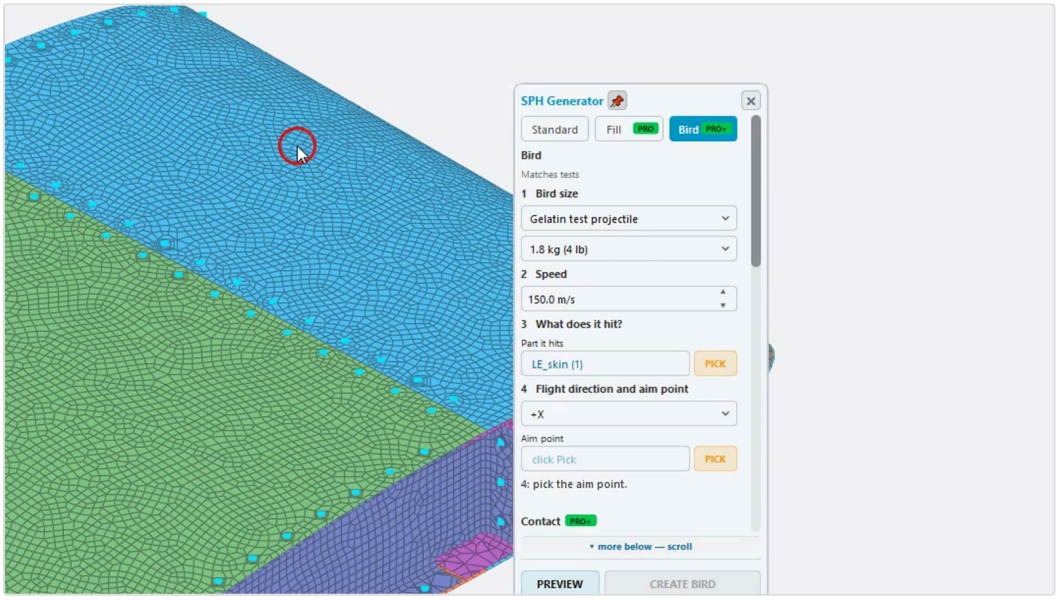
Step two is the speed.

1. Type one hundred fifty metres per second.

Step three: what does it hit?

2. Press Pick, and click the leading edge skin.

In the video: **150** in Speed.



The screen at 0:39 in the video.

4 Direction, Aim, Hits line

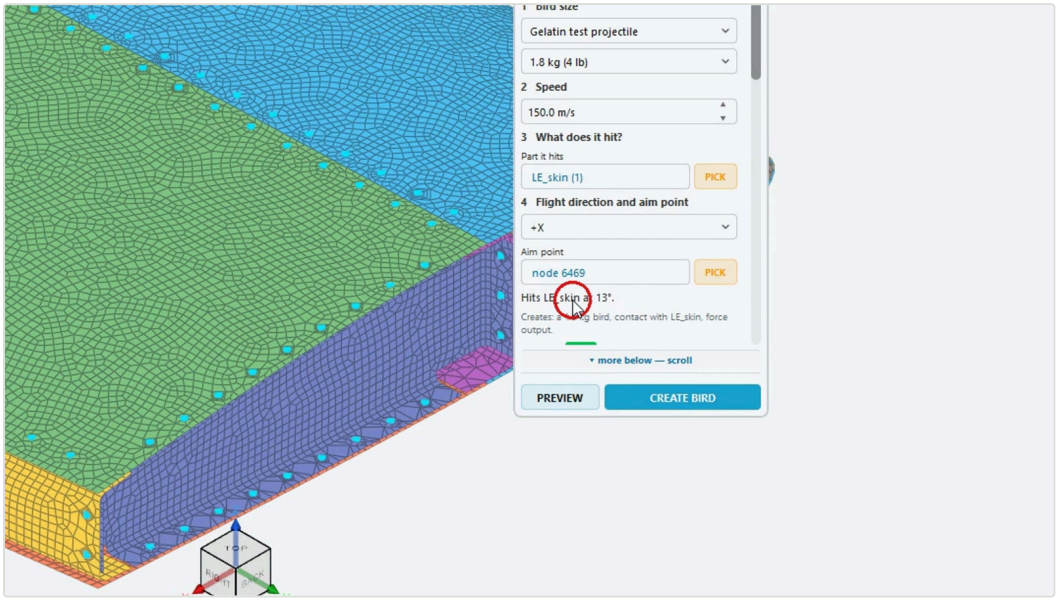
video 0:39

Step four is the flight direction and the aim point.

The bird flies along plus X.

1. Press Pick, and click the node to aim at.

   The panel now says what the bird hits first, and at what angle.



The screen at 0:51 in the video.

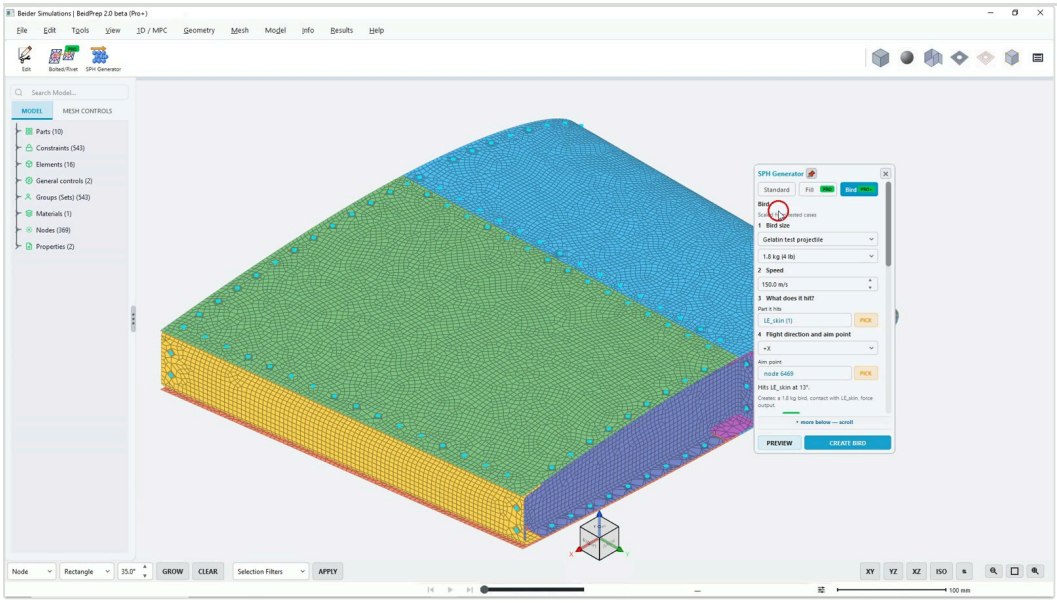
5 Summary, Status

video 0:51

It also lists what Create bird will make.

The status now says Scaled from tested cases.

The bird is set up from published bird strike tests, scaled to this case.



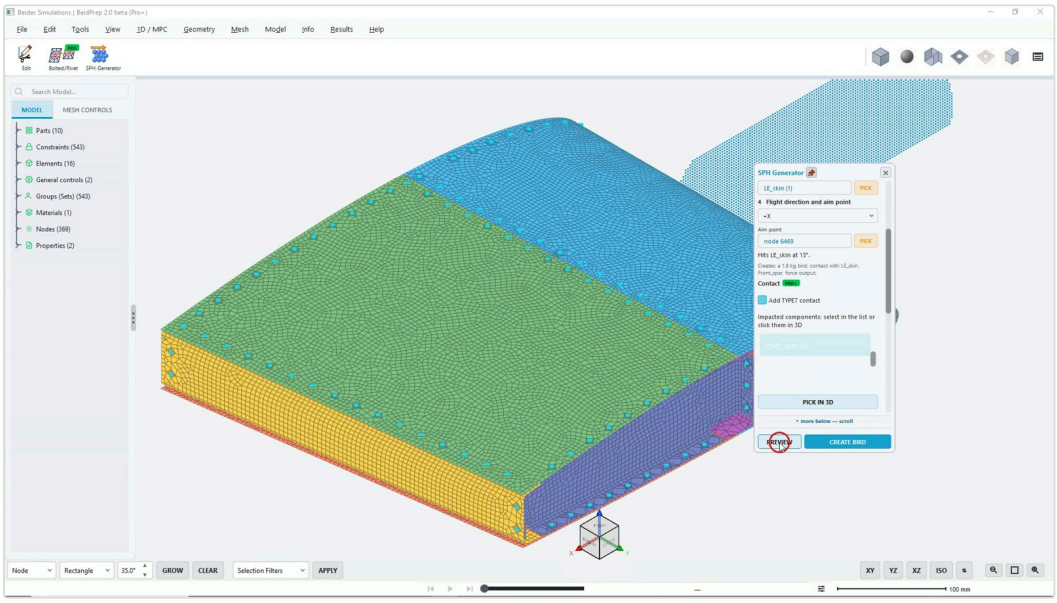
The screen at 1:01 in the video.

6 Contact, Preview

video 1:02

The contact is already on, with the leading edge skin in it.

1. Add the spar, which the bird can reach behind the skin.
2. Press Preview to see the bird in the 3D view.



The screen at 1:12 in the video.

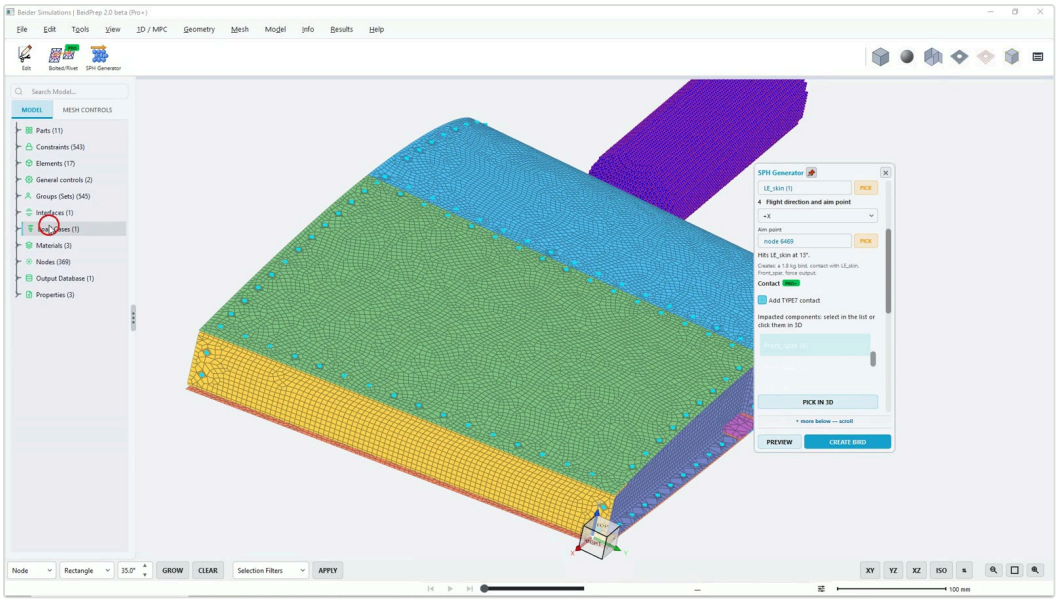
7 Gap, Create, Tree

video 1:13

The bird is placed just before the first surface it would touch, so it never starts inside a part.

1. Press Create bird.

The bird and every card it needs are now in the model tree.



The screen at 1:23 in the video.

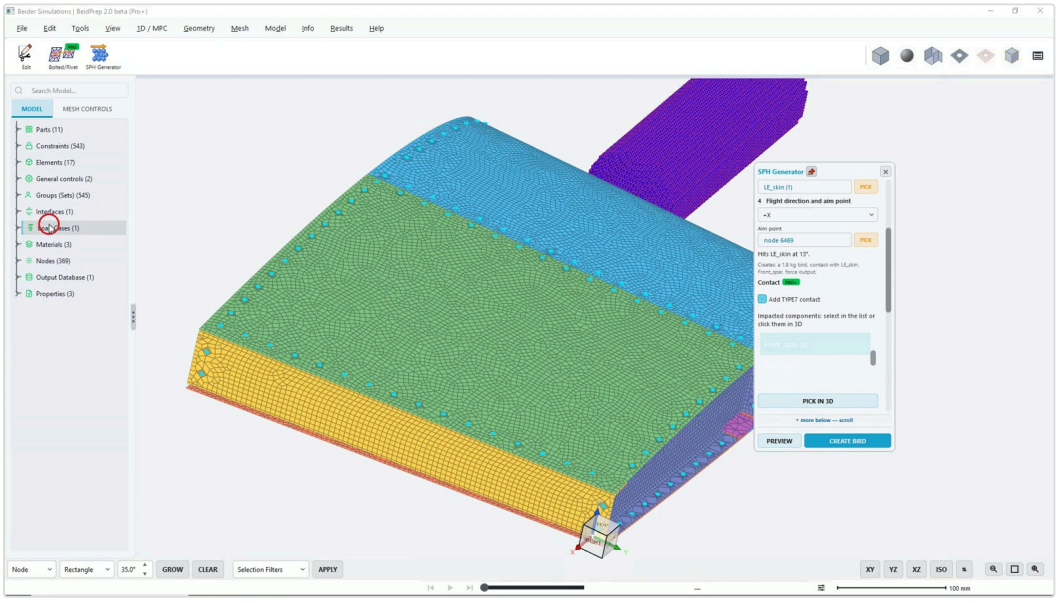
8 Undo, Close

video 1:23

One Create bird is one step.

Control Z removes the bird and all its cards.

That is the Bird tab: size, speed, target and aim, in four steps.



The screen at 1:26 in the video.

Steps and pictures are taken from tutorial video 13. Every picture is a frame of the video.

TUTORIAL VIDEO 14

14 Working with Claude: Claude Code on the model you have open

With Pro+, Claude Code on your own computer works with the model you have open in BeidPrep: choose what it may read and change, ask about the model, and approve each change -- every request is logged, and each change can be undone.

Tutorial files	folder `14_Working_with_Claude` — open `s_rail_0000.rad` first
Video	Watch video 14 on YouTube
Download files	Download the tutorial files (97 MB)

1 Opening, Connect window, What Claude may read and change

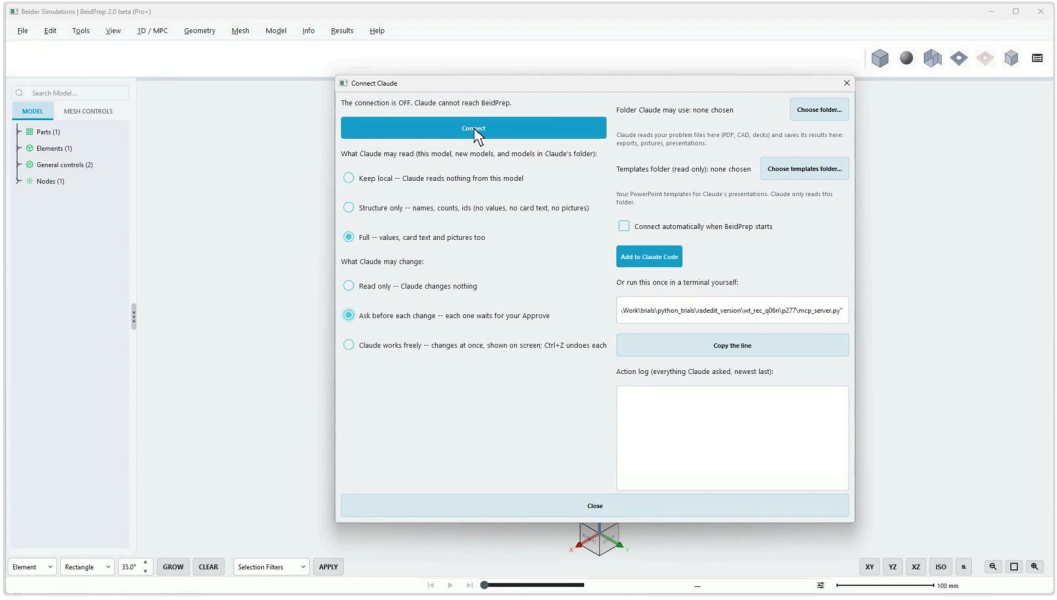
video 0:00

With Pro plus, Claude Code on your own computer can work with the model you have open in BeidPrep.

1. Choose Connect Claude in the Tools menu.

   First, what Claude may read: Full lets it read values and card text too.

   Then what it may change: with Ask before each change, every change waits for your Approve.



The screen at 0:19 in the video.

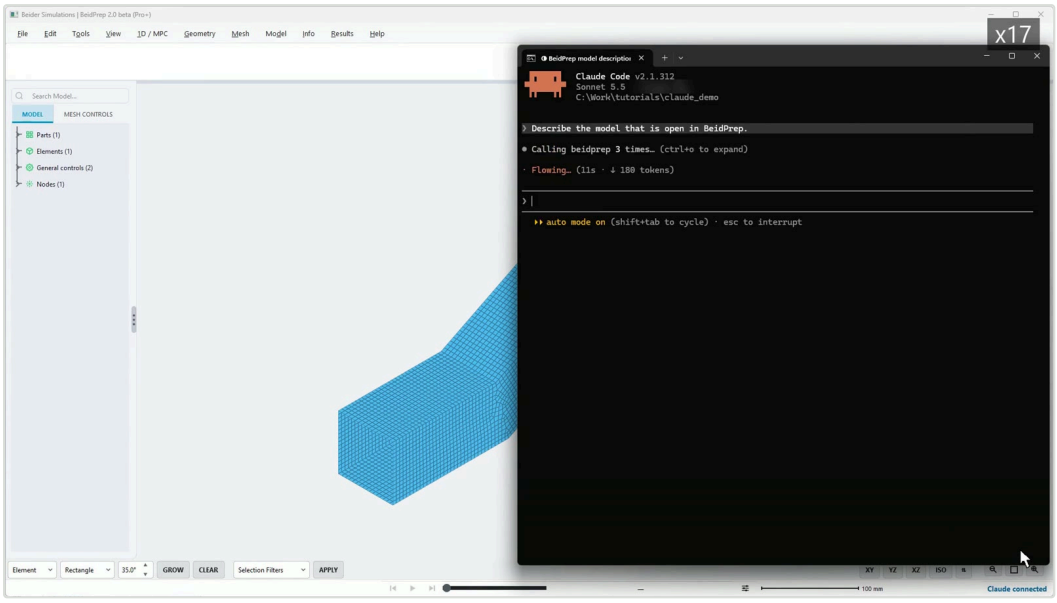
2 Connect, Ask about the model

video 0:19

1. Press Connect. The connection stays on this computer, with a new key every session.

Claude Code runs in a terminal, in your model's folder.

2 Ask it about the model you have open.



The screen at 0:33 in the video.

3 The answer, Ask for a change, Approve

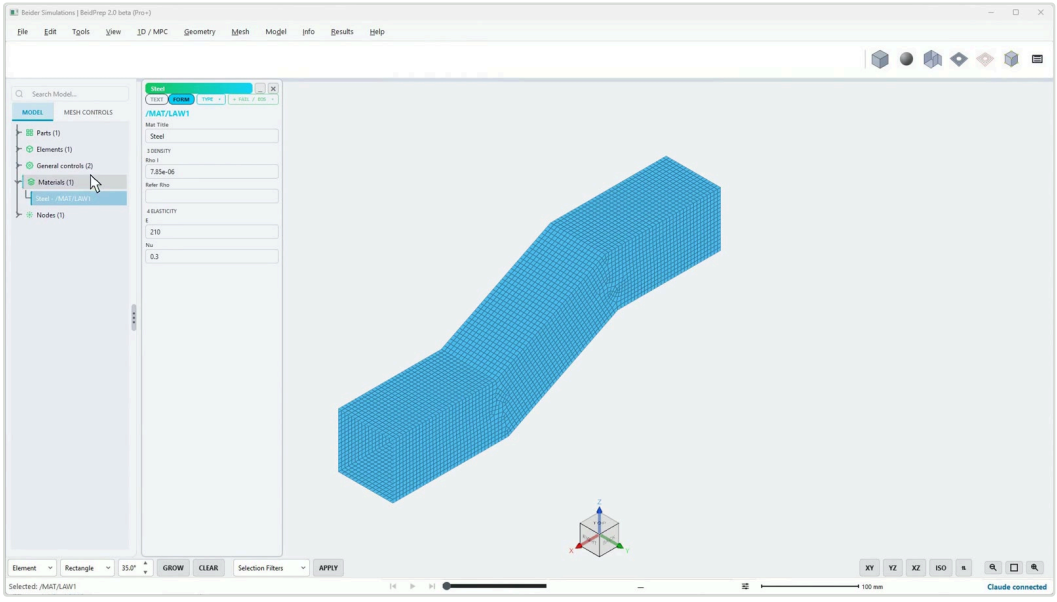
video 0:34

Claude reads the model through BeidPrep, and answers from what it finds there.

1 Now ask for a change, and give it the values to use.

BeidPrep shows what Claude wants to do, and waits: press Approve.

The new material is in the model tree, like any card you make.



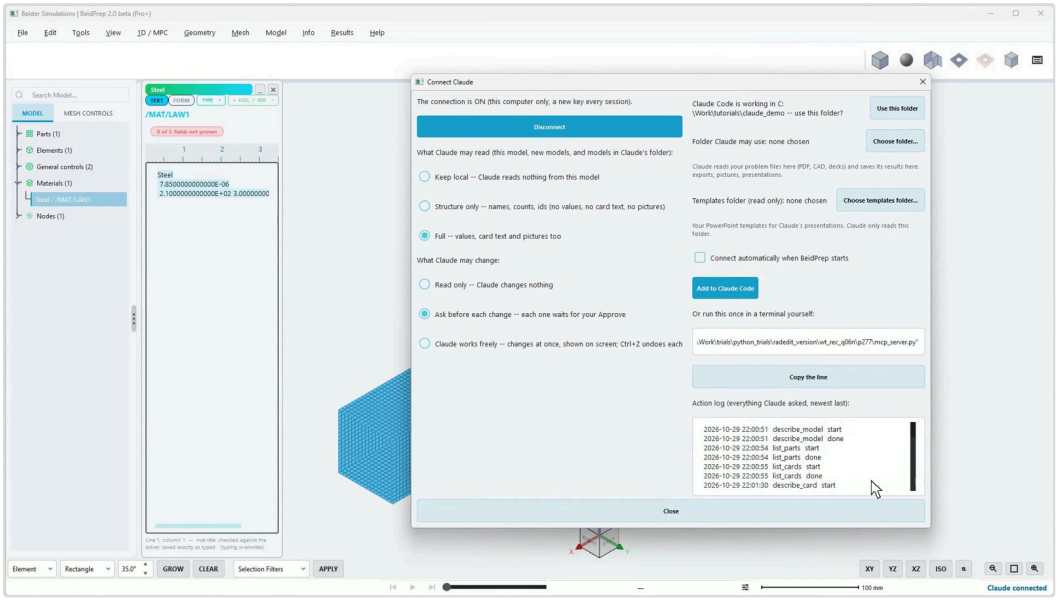
The screen at 0:51 in the video.

4 Close

video 0:53

With Claude works freely, changes happen at once, and Control Z undoes each one.

Every request Claude made is listed in the Connect window's action log.



The screen at 1:03 in the video.

Steps and pictures are taken from tutorial video 14. Every picture is a frame of the video.