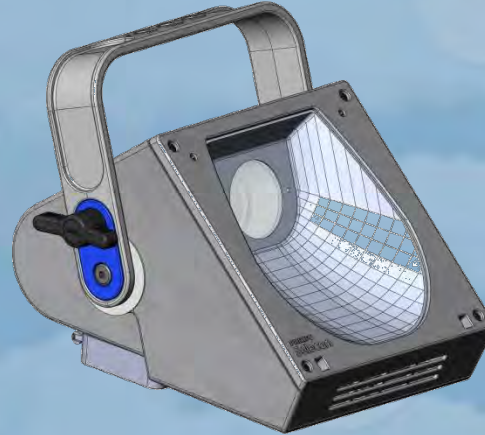# Practical Plastic Part Design

Lindsay Dalziel
**Design Engineer**

www.DesignSMART.co.nz

Tips & techniques for Plastic Part Design

- **SolidWorks Tips & other practical advise:-**
  - Plastic Part (Mould) Draft analysis and how much.
  - Add Mould Draft to features at creation where possible.
  - Best to Mirror Bodies before applying the Draft Features.
  - Select from upstream sketches rather than part edges
  - Add the Fillets last, largest to smallest etc.
  - Early & effective communication with Toolmaker is key.
  - The use of Sliding Cores / Wedges.
  - Gating the Part / Mould & effects.
  - Shrinkage / off tool warp-age of Parts.
  - Rapid Prototyping and or Analysis
  - High Quality Rapid Prototyping parts.
  - Warp-age of Parts when in use.
  - Over Moulding.
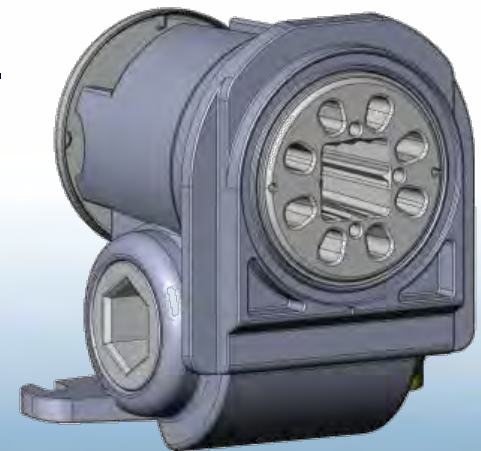
Other Tips & techniques for Part Design

- **More SolidWorks modeling techniques:-**

  - Use high quality construction (sketch) geometry

  - Keep (upstream) construction geometry clean & simple

  - Preserve Sketch data, convert to construction if unsure

  - Use Surfaces to aid construction of Solids

  - Use of strategically placed surfaces in the tree

  - Copying surfaces across parts in an assembly

- **If there's time?:-**

  - The value of using Layout Parts for construction

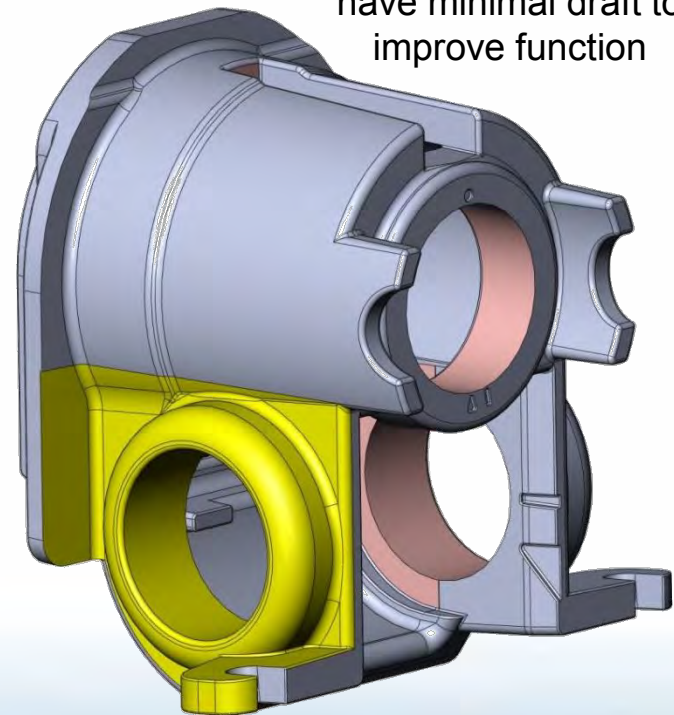 - Dealing with two physical states of a part
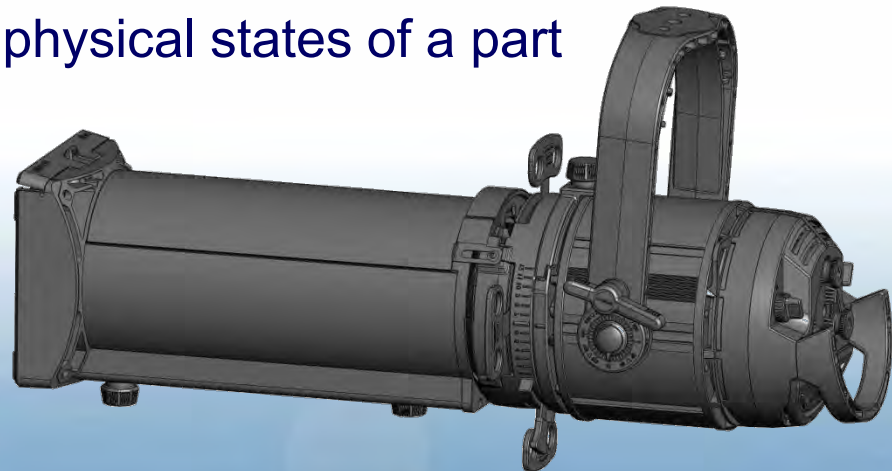
Draft analysis, Technical Moulding

- **Set up a hot key, and use it often, (e.g. F8).**
- **Rough Rule 2 degrees on internal, 1 degree on external faces.**
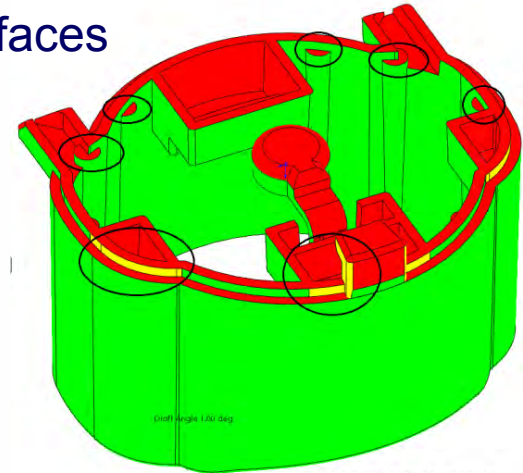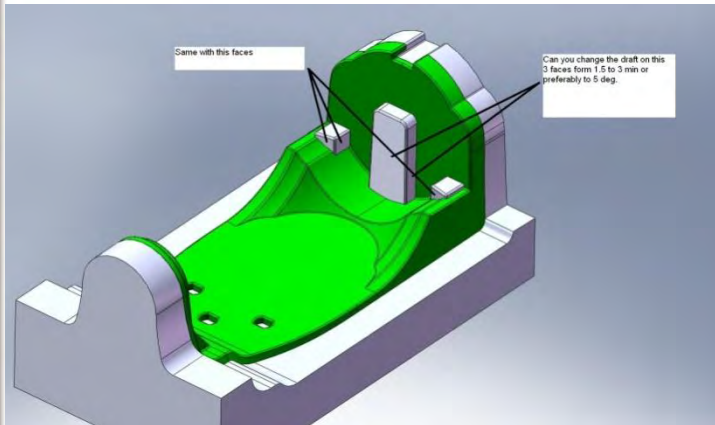 - Depends on size & depth of face
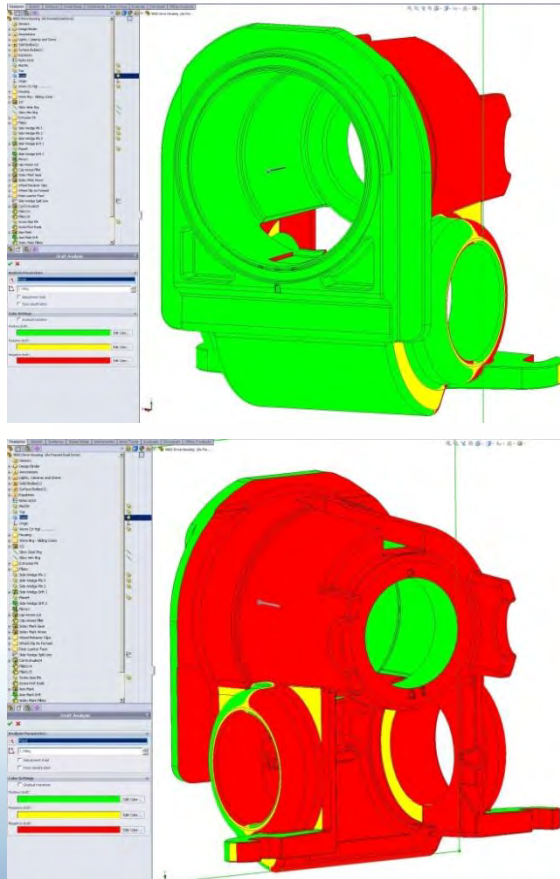
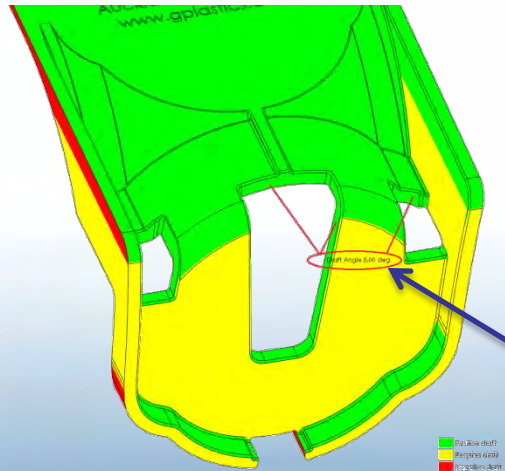Pink Coloured Internal faces have minimal draft to improve function

Yellow Coloured faces Formed by Sliding Core on Both Sides

More Mould Draft

- **Bursting through from core to cavity, to form apertures.**
  - 3 to 5 degrees draft on specified faces, as below left.
  - Again Depends on size & depth of faces

Can you change the draft on this 3 faces form 1.5 to 3 min or preferably to 5 deg.

Same with this faces

I think this model is rectifiable, just need to confirm the Moving / Ejector Pin Side.

Tool maker communication best done with screen shots, and MS Paint markup, everyone can do it, even technophobes, its quicker, & smaller file size than eDrawings i.e. suitable for emailing.

5 Degree Draft applied to these faces, otherwise you have a Morse Taper effect in the Tool shut off.

Draft Increased to 5 degrees
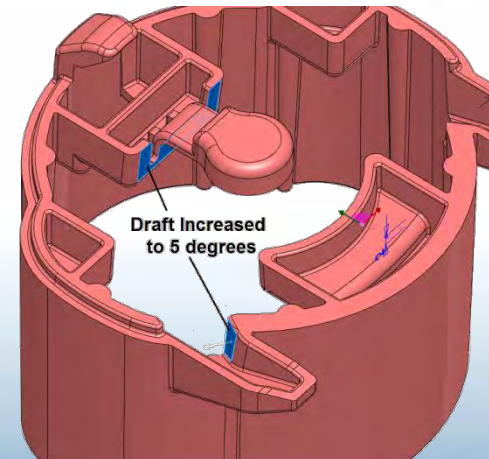
Blue Coloured faces have shut off edges Both normal to & in line of draw hence needed more draft

©2012 Design Smart Ltd

Adding Draft

- **Add Mould Draft to features at creation where possible**
  - Keeps the feature tree more compact, & the geometry is more stable.
  - Best to mirror Bodies then apply independent Draft features later.
  - Tricky independent Draft features frequently flip when mirrored.
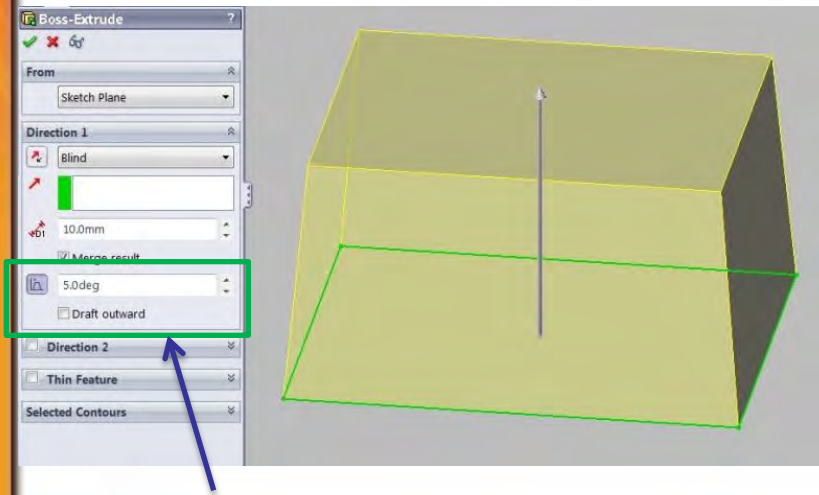
Add Draft within the feature where practical for the most stable geometry
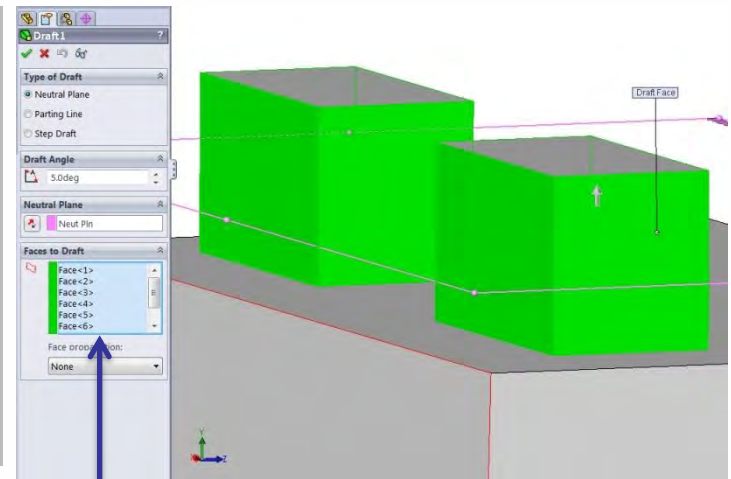
Draft feature applied to Independent faces has more options but can be less stable, so if the functionality is not needed put the draft into the Feature at birth.
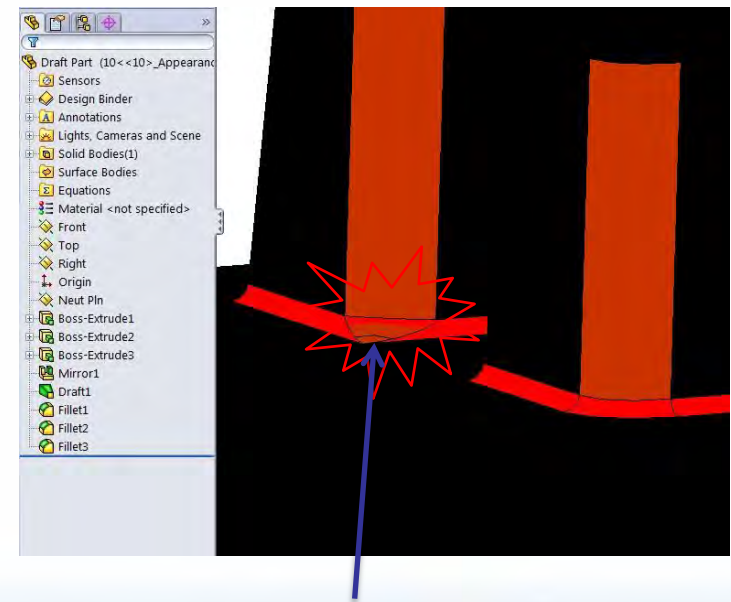
Adding Fillets

- **Add Fillets last, in order of largest to smallest fillets**
    - This generates best quality fillets for N/C machining, also you have the best chance of getting a result with the automatic tangent selection tracking, i.e. less mouse selection.
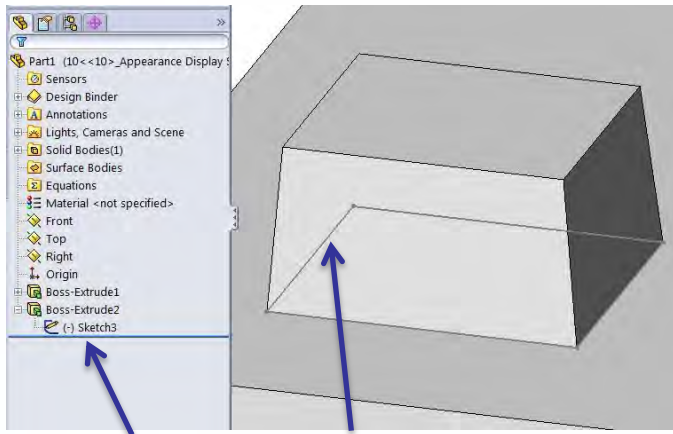

Not Good for NC

Good Result
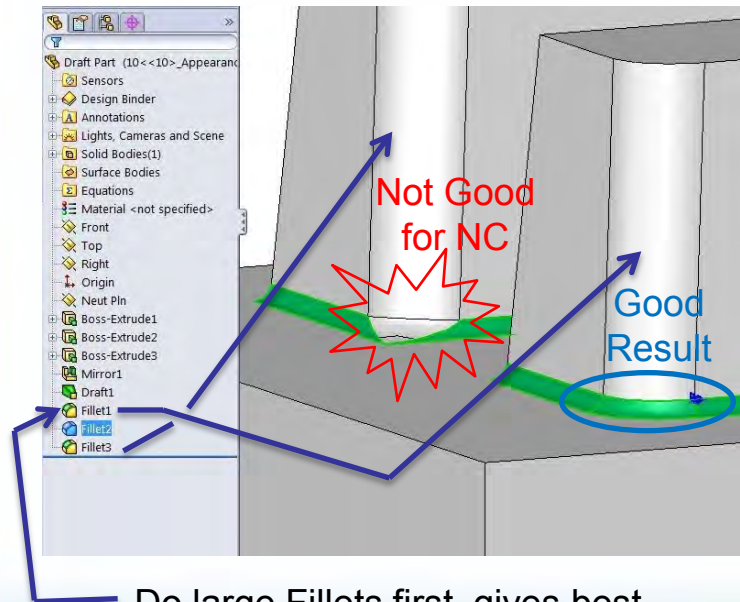
Do large Fillets first, gives best result and best chance of a result.
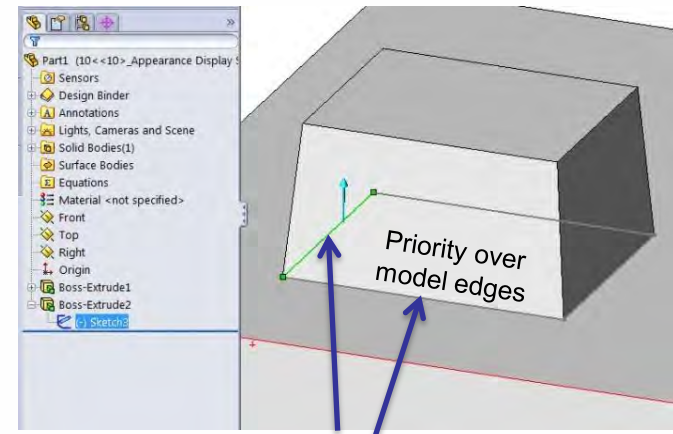

Variable Curvature result in this external suitcase corner makes for a difficult NC tool path.
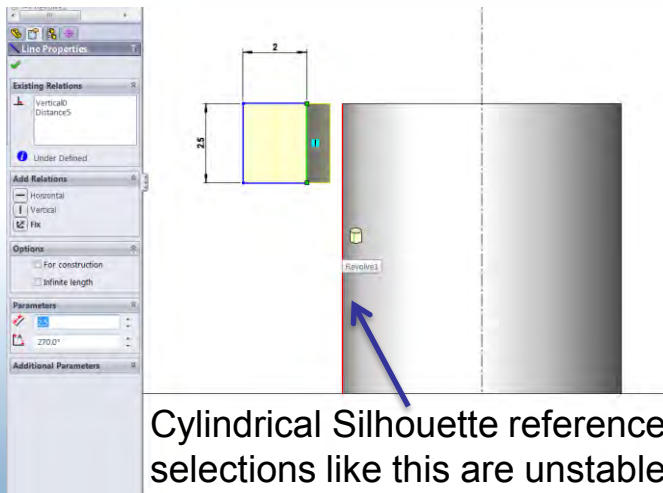
Reference Geometry Selection

- **Select the upstream sketch geometry for max stability**
  - Once the upstream sketch is shown, it gets priority.
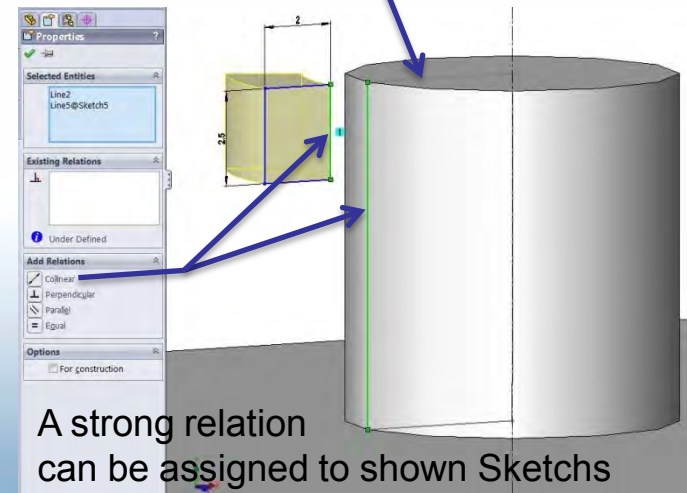

Right click, to Show Sketch's for selection

Shown Sketch entities get selection priority

Priority over model edges

Cylindrical Silhouette reference
selections like this are unstable

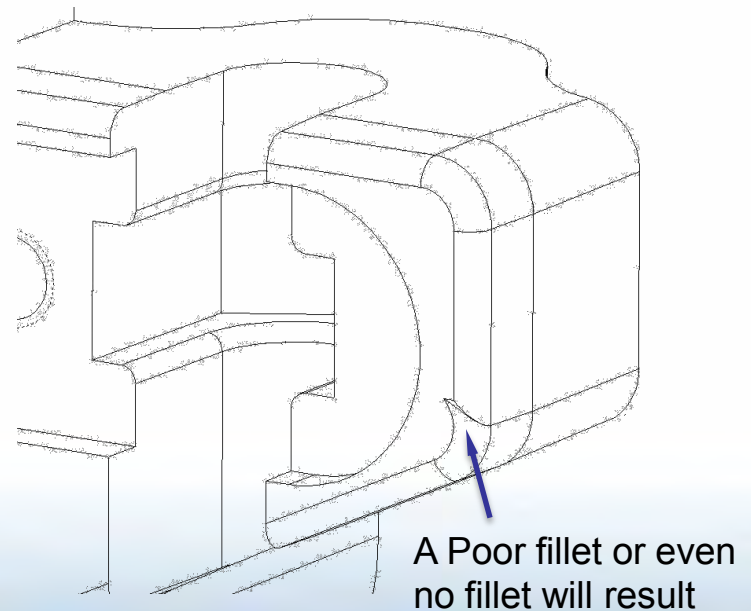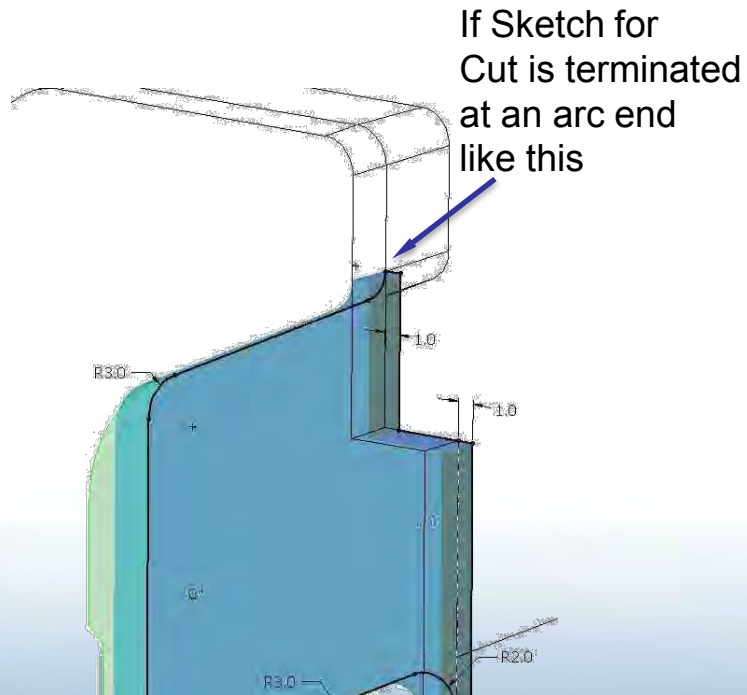A strong relation
can be assigned to shown Sketchs

Fillet Quality

- **Quality of the upstream geometry is key**
  - An example is if the previous features in the model includes draft, watch exactly where upstream cuts are terminated, i.e. don't finish sketch's at the end of an arc as below.

If Sketch for Cut is terminated at an arc end like this

A Poor fillet or even no fillet will result

For Quality Filleting

- **The upstream geometry is everything**
    - When draft is included in features finish your sketch for cuts beyond arc ends, dimension to the arc end if possible, and finish on a line entity in the sketch if possible.
      Good fillets result, see below.

So Finish Sketch for the Cut up here on a Line

Go past arc end if possible

Good Corner fillet results
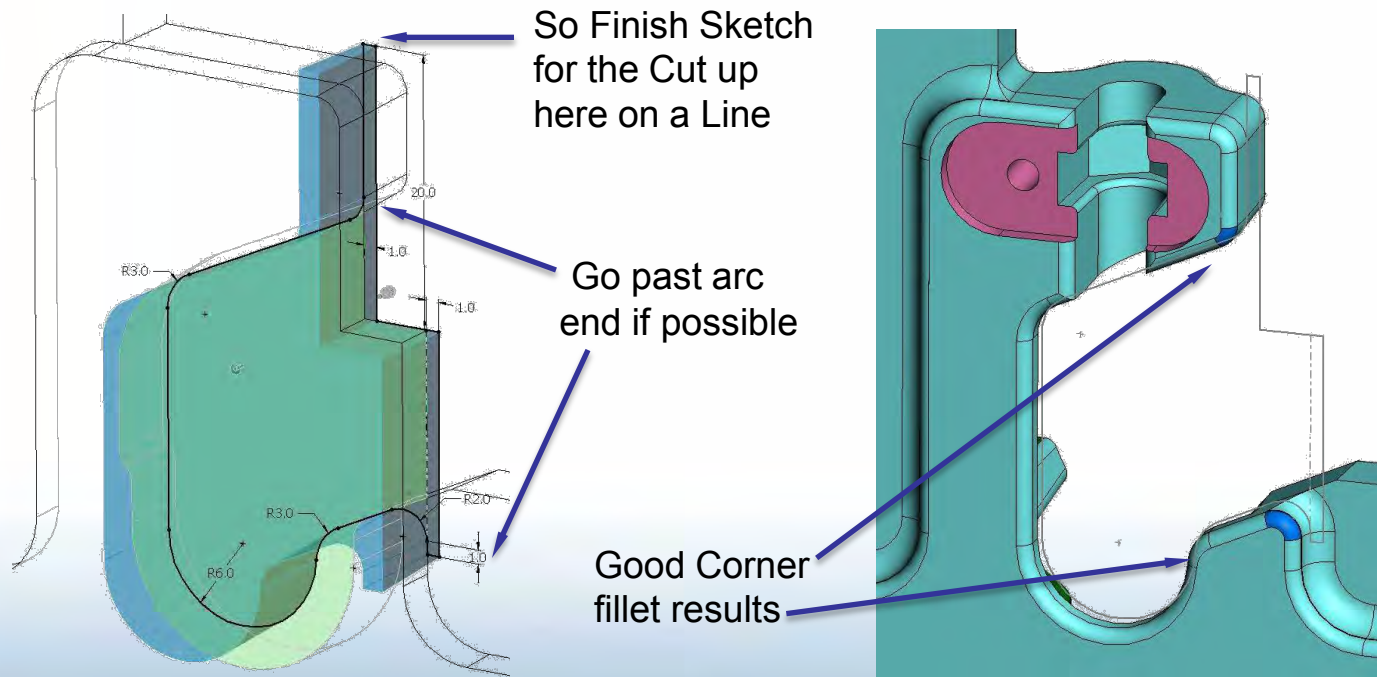
Sliding Cores / Wedges

- ## For Simplifying part count & assembly
  - And also improve Structural integrity of the Enclosure

Case

Front Cover

Electronics to be enclosed

X-Sec showing placement of parts
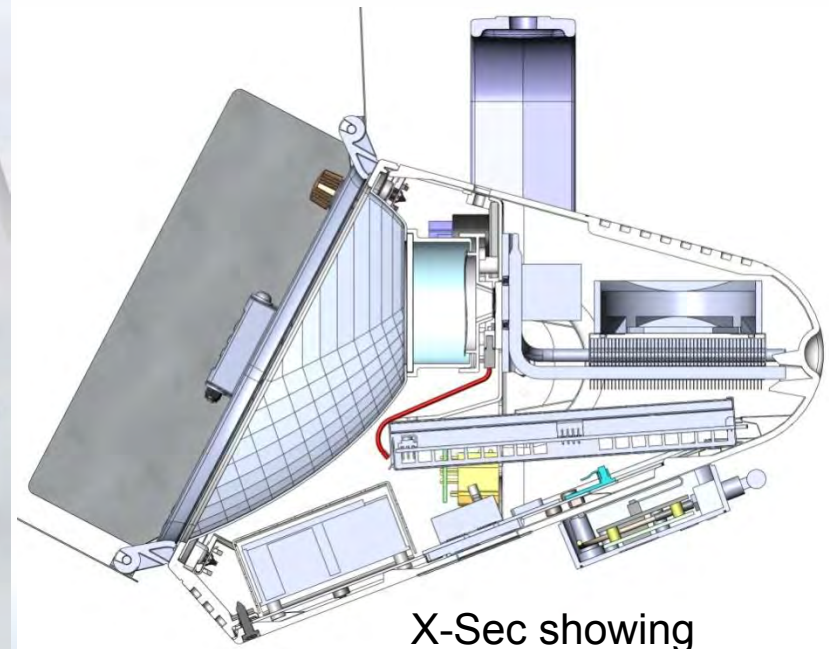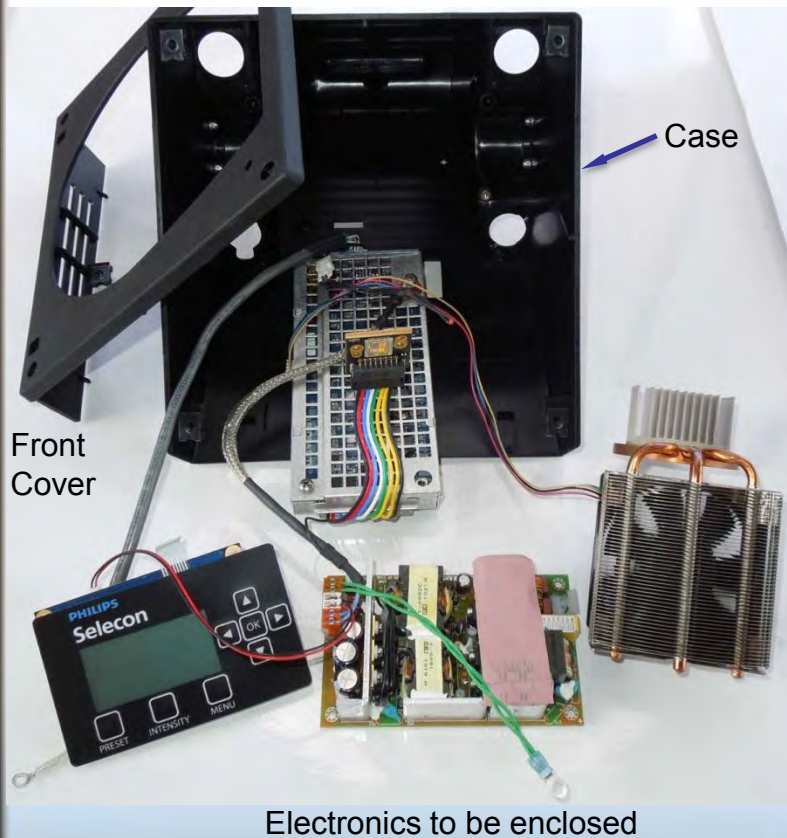
Sliding Cores / Wedges

- ## Simplifying part count & assembly
  - Used for Fastening of the Front Cover to the Case
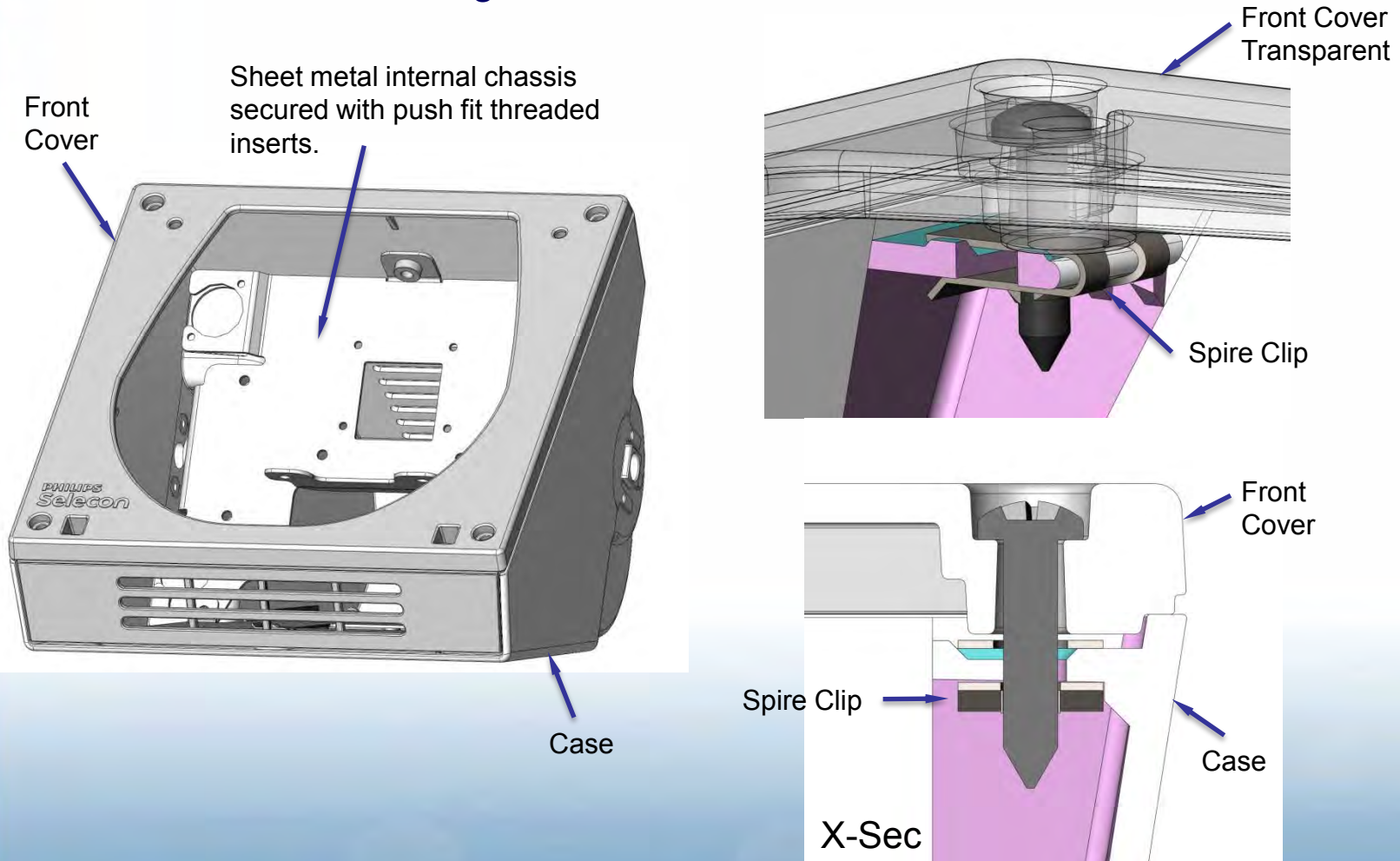
Front Cover

Sheet metal internal chassis secured with push fit threaded inserts.

Case

Front Cover Transparent

Spire Clip

Front Cover

Spire Clip

Case

X-Sec

Sliding Cores / Wedges

- ## Simplifying part count & assembly
  - Using a Wedge within a Wedge to do the Top Corners
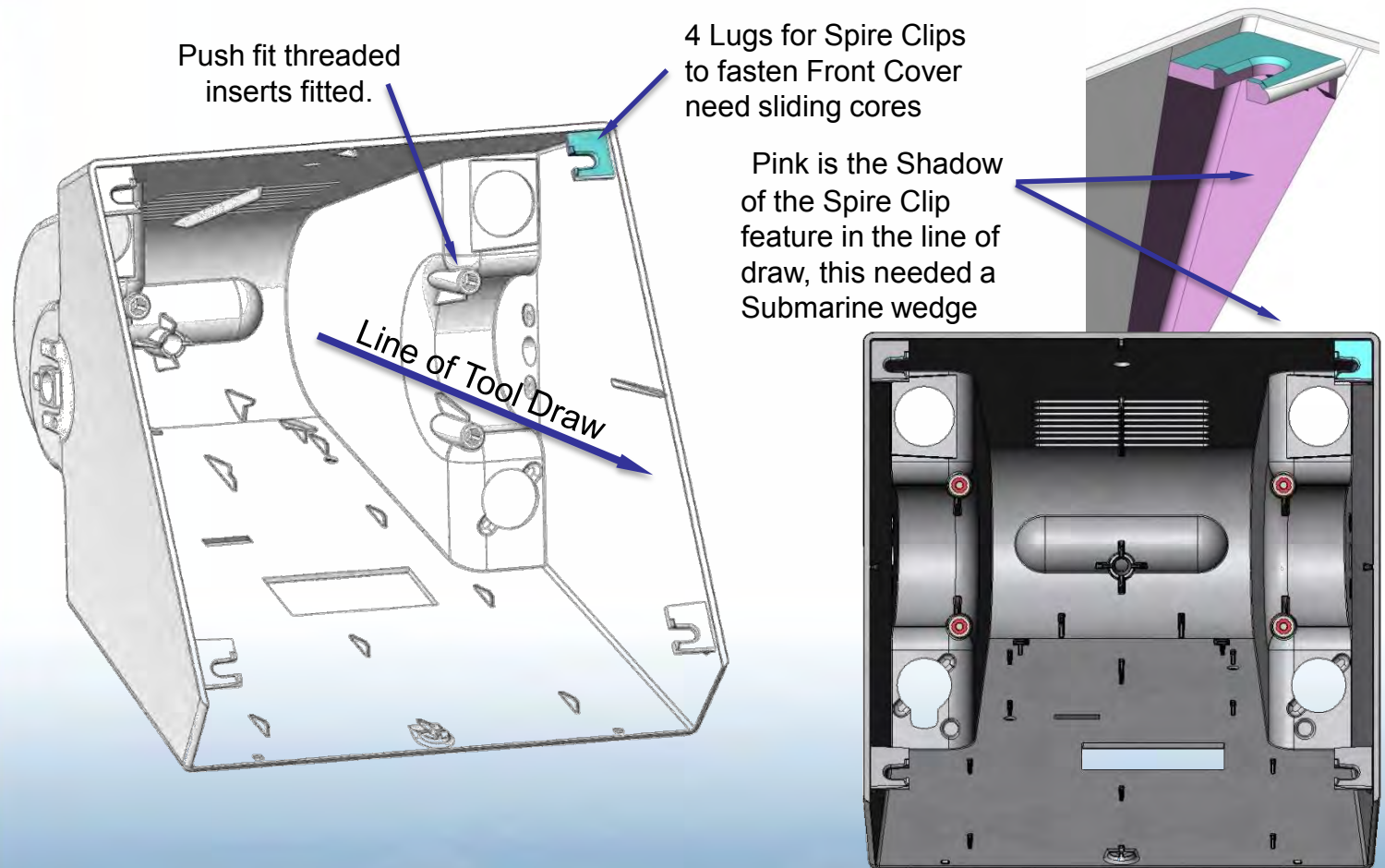
Push fit threaded inserts fitted.

4 Lugs for Spire Clips to fasten Front Cover need sliding cores

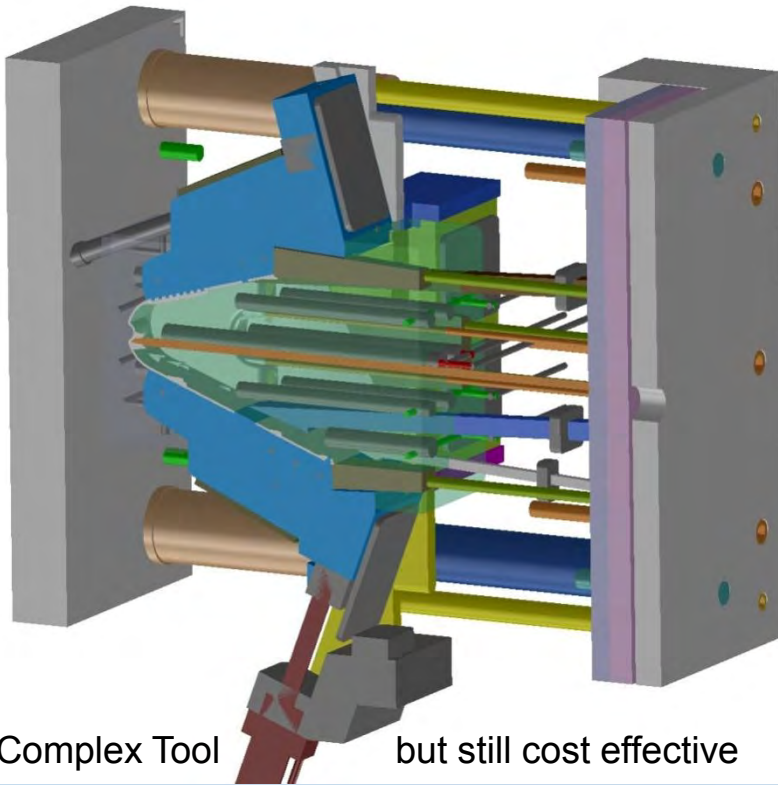Pink is the Shadow of the Spire Clip feature in the line of draw, this needed a Submarine wedge
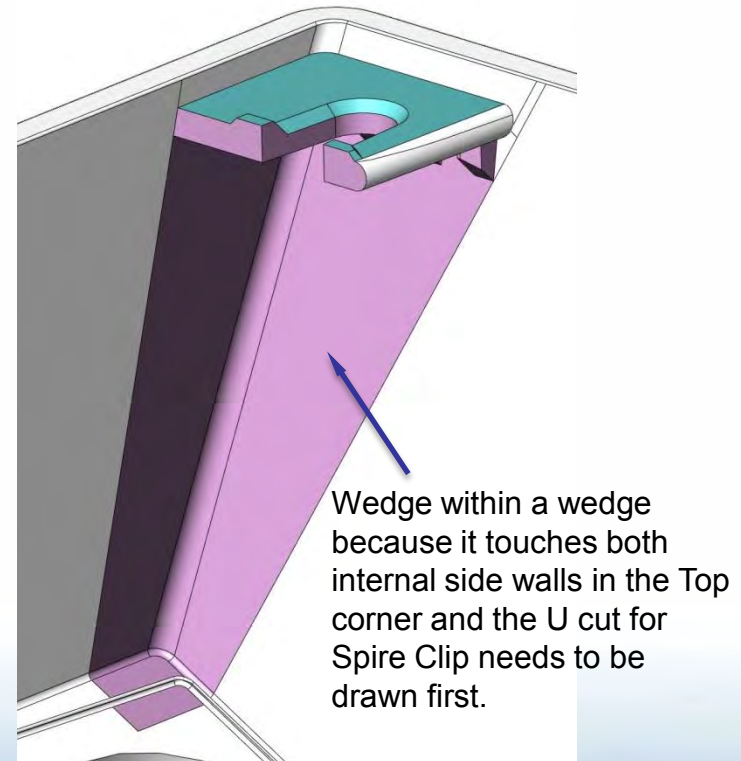
Line of Tool Draw

Sliding Cores / Wedges

- ## Simplifying part count & assembly
  - ### Using a Wedge within a Wedge to do the Top Corners

Complex Tool   but still cost effective

Wedge within a wedge because it touches both internal side walls in the Top corner and the U cut for Spire Clip needs to be drawn first.
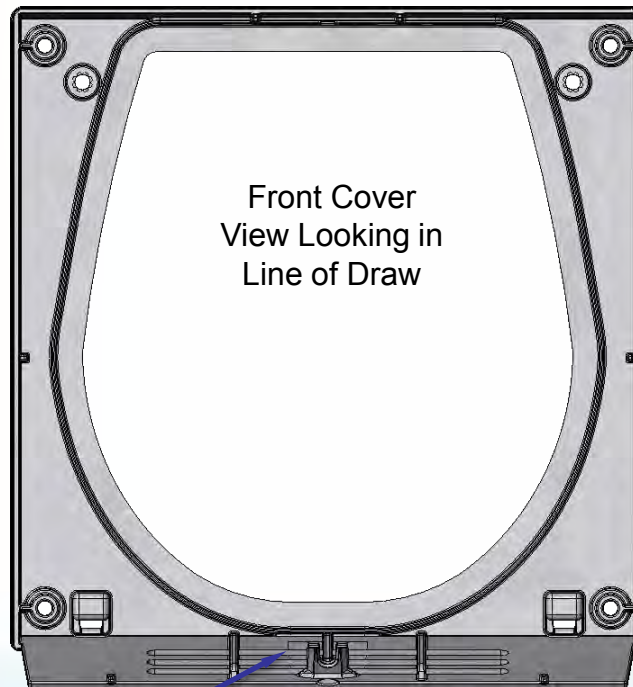
Alternative would be using 2 parts which = 2 Tools
No cheaper and more assembly time required
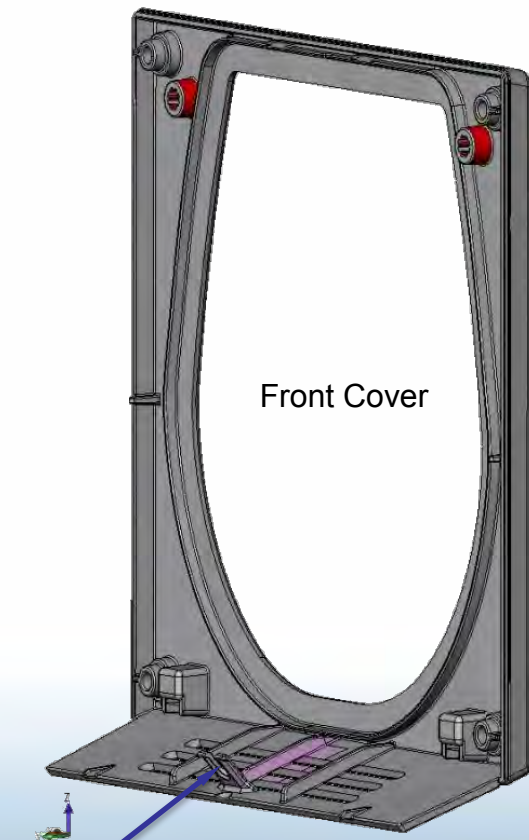And the Cases Structural integrity not as strong

Sliding Cores / Wedges

- ## Simplifying part count & assembly
  - Simple Wedge to completely tie Case & Front Cover together

Front Cover
View Looking in
Line of Draw

Front Cover

Pink face is in the Shadow of the Spire Clip feature in the line of Tool draw, this result was achieved with a Simple wedge, as easy Tooling access to the required geometry
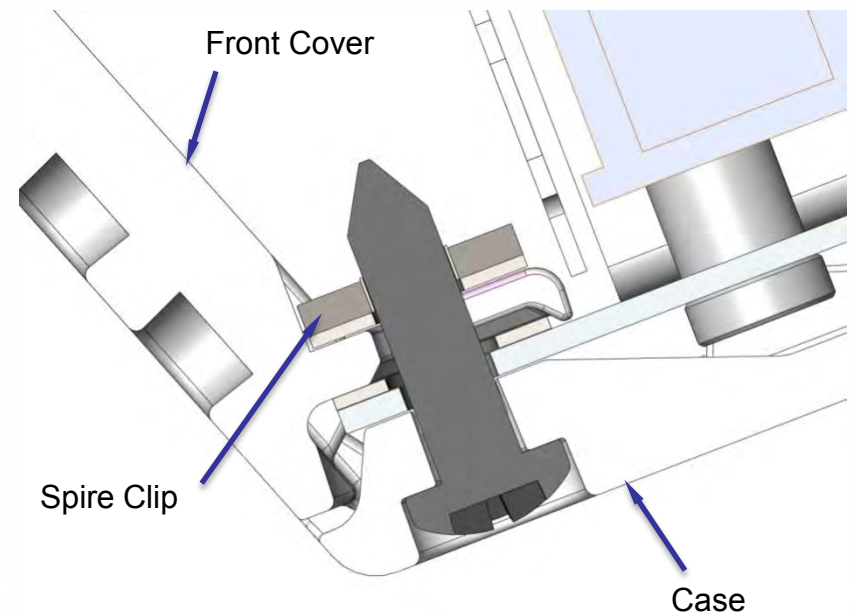
Spire Clip feature

Sliding Cores / Wedges

- **Simplifying part count & assembly**
 - Simple Wedge to completely tie Case & Front Cover together

Front Cover
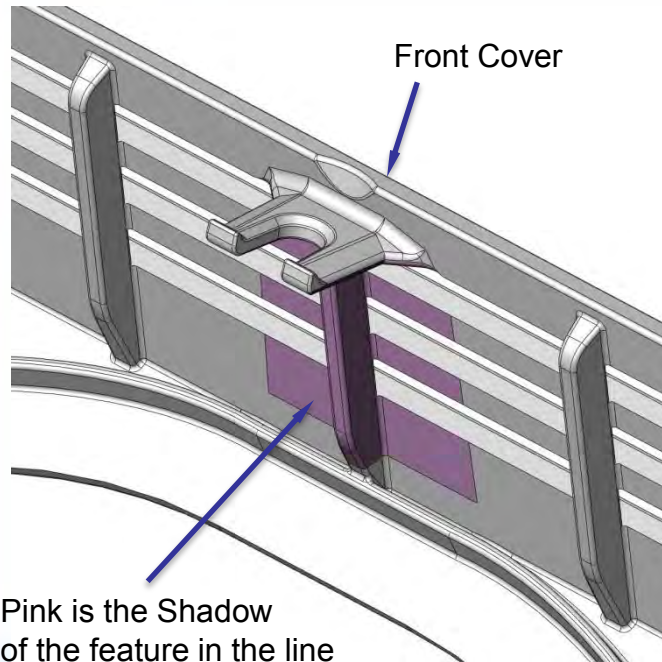
Pink is the Shadow
of the feature in the line
of draw, this result achieved with
a Simple wedge with easy
access to the required geometry

Front Cover

Spire Clip
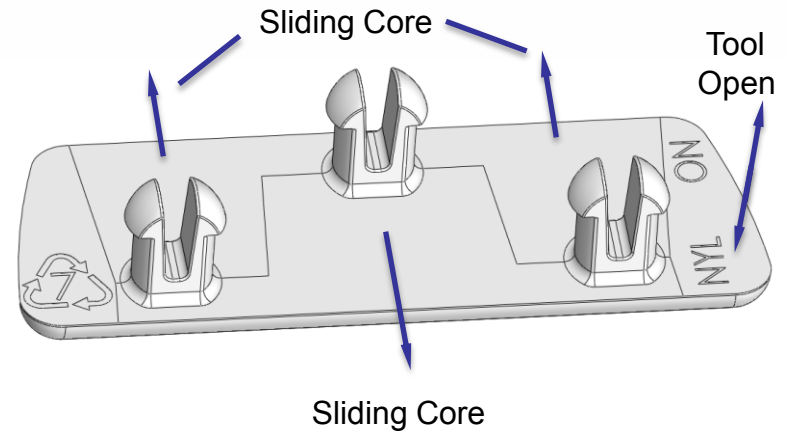
Case

Central X-Section Bottom Front of Assembly

Sliding Cores

- ## To Create a Gang Clip Part
  - First thought was open Top to Bottom
  - Front face of Part (not shown) needed embossed Branding
  - So Tool Maker suggested Wedges & open Front to Back

Positive draft
Requires draft
Negative draft
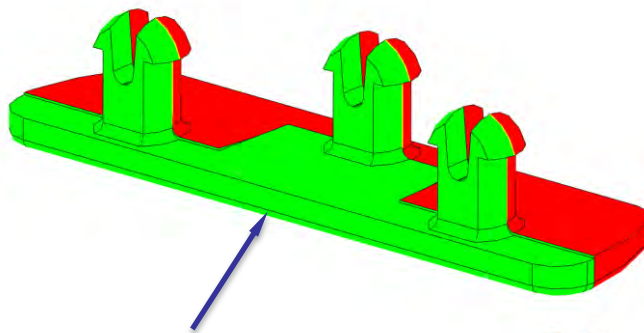
First Thoughts were to
Open the Tool Top Bottom
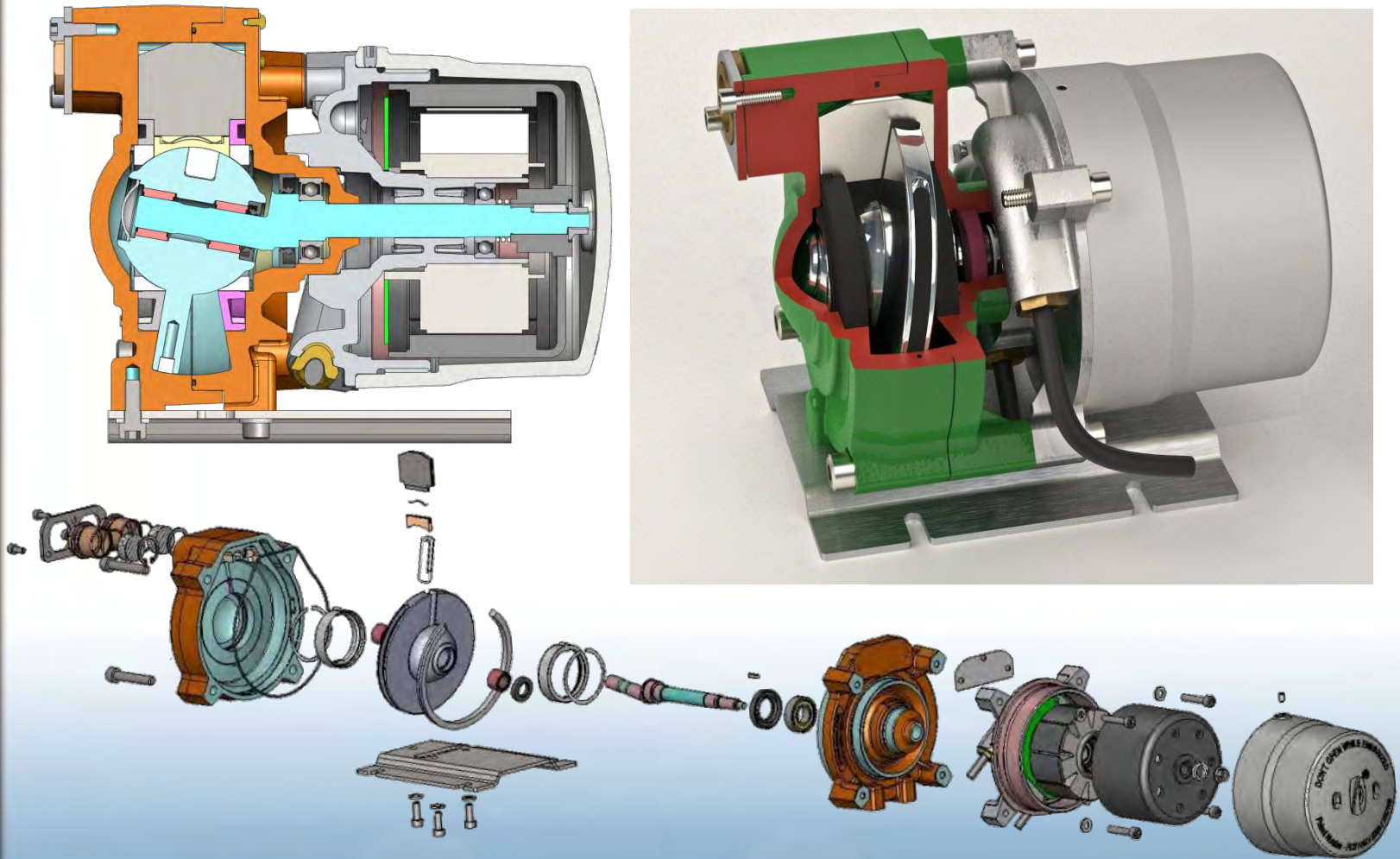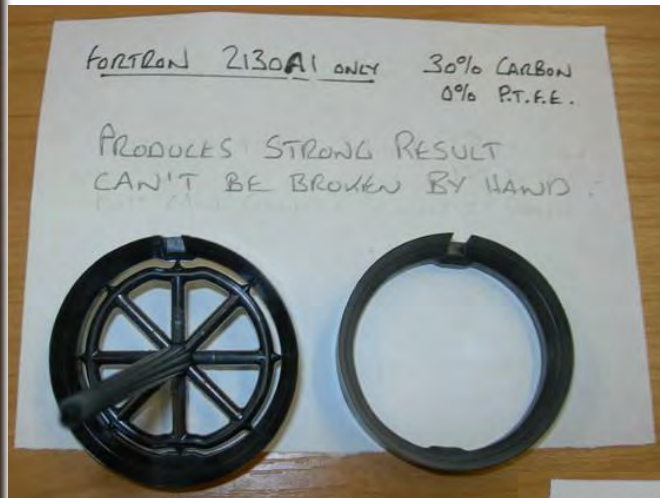

Sliding Core

Tool Open

Sliding Core

Gating of the Parts & effects

- Gating of Plastic Tools is important to the final shape & strength of parts, the Vapour Pump project has good examples of this:-

Gating of the Tool & effects

- **Gating of parts is important for shape & strength**
  - Also effected by additives, especially PTFE

Result when 8 Point Sprue Gated, strong result above. Note the roundness & diameter size of this Part is important Plus Minus 0.05mm tolerance

Small amounts of PTFE Additive made for strange results prior to the full Fan Gating of the Tool

Note:- We also had Batch variance diameter size issues with this part USA Sourced material v Malaysian

Gating and Warpage of Parts

- **Gating of parts is important for Warpage & strength**

Sealing Ring:-, as above A part shaped like this will always pull in on diameter regardless of the injection point / gating, Final Tool Diameter was 117.15mm O.D. for an 115.30mm O.D. off Tool result. Was injected at one end, which produced acceptable Roundness
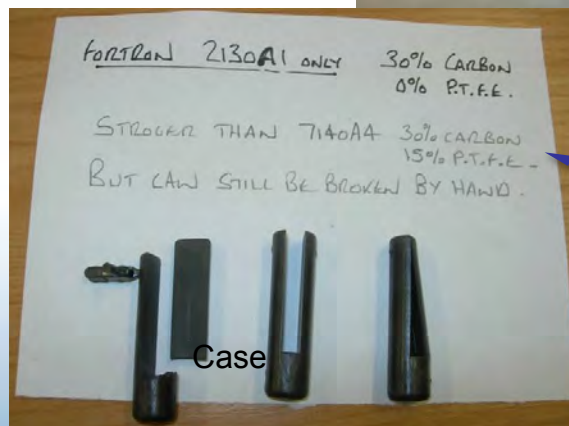
Trunnion Bearing Part:-

Bottom Side Gated Result splayed out at the Top

Top End Gated Result Splayed In at Top

Finally Ring Gated at the Bottom for a good result

FORTRON 2130A1 ONLY 30% CARBON 0% P.T.F.E.

STRONGER THAN 7140A4 30% CARBON 15% P.T.F.E.

BUT CAN STILL BE BROKEN BY HAND.

Case

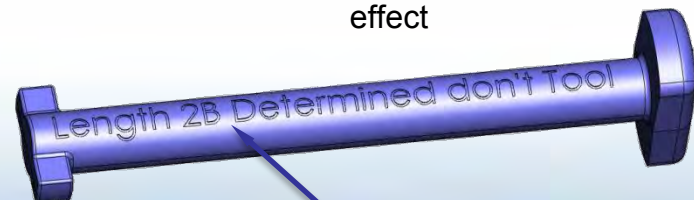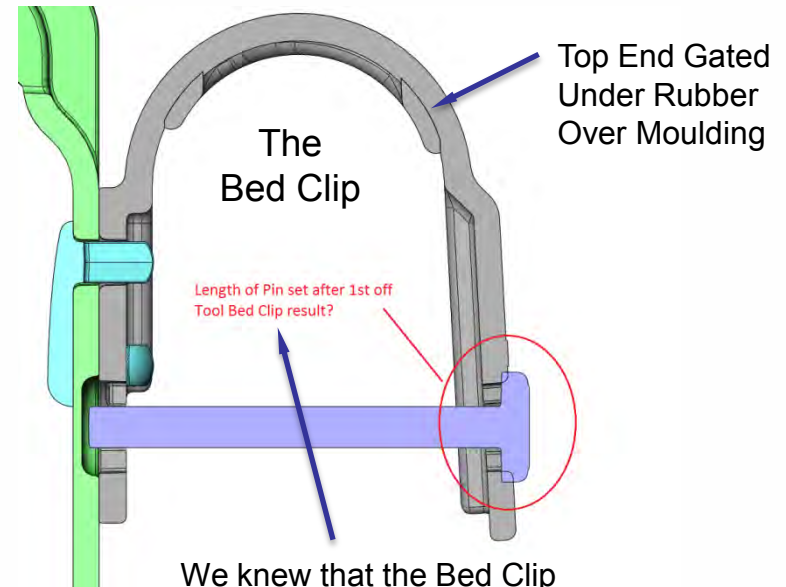PTFE Additive again made for strange results

More on Warpage of Parts

- **Warpage can be surprisingly excessive:-**

Bottle Holder for Hospital Beds

NYLON

Manufactured by
Galantai Plastics Group Limited
Auckland New Zealand
www.gplastics.co.nz

The
Bed Clip

Top End Gated
Under Rubber
Over Moulding

Length of Pin set after 1st off
Tool Bed Clip result?

We knew that the Bed Clip
was going to Pull in (off Tool)
and notes were made to this
effect

Length 2B Determined don't Tool

Lock Pin

Communication even set
in 3D into part data, as no
Drawings, STEP files only

Rapid Prototyping & or Analysis

- **The Folding Head Band, example:-**
  - Geometry for the Band requires a hole for molding purposes to achieve the swivel lock feature
  - The concern then was the relative strength of the band's different cross sections and swivel clip.

Hole required for molding

Swivel clip rotates & retains

Consistent strength of band sections required
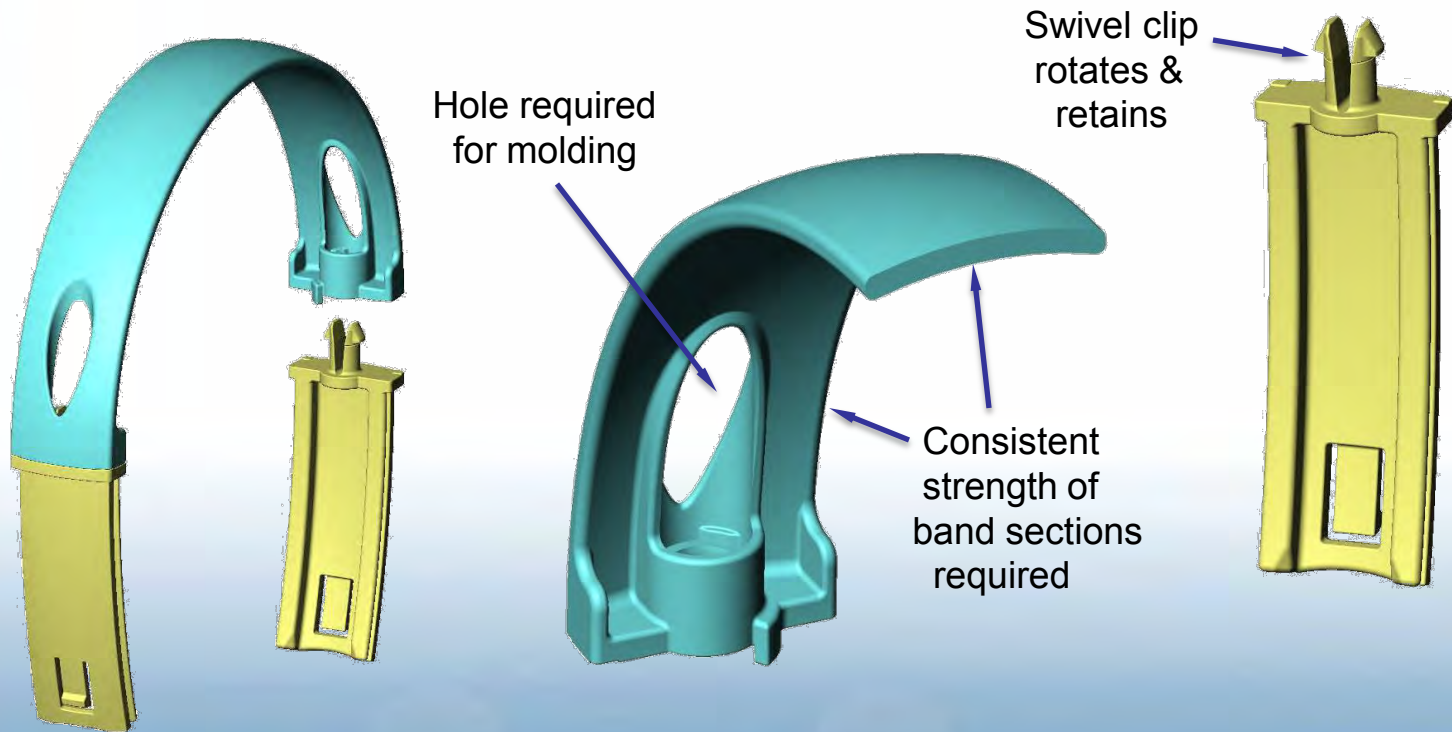
Folding Headband analysis

- ## A strength analysis conducted

  - Out of curiosity this analysis was done at the same time rapid prototypes were being made, was done by a guy at Cosmos

 - This raised concern suggesting clip failure before any significant Headband deflection!

Stress
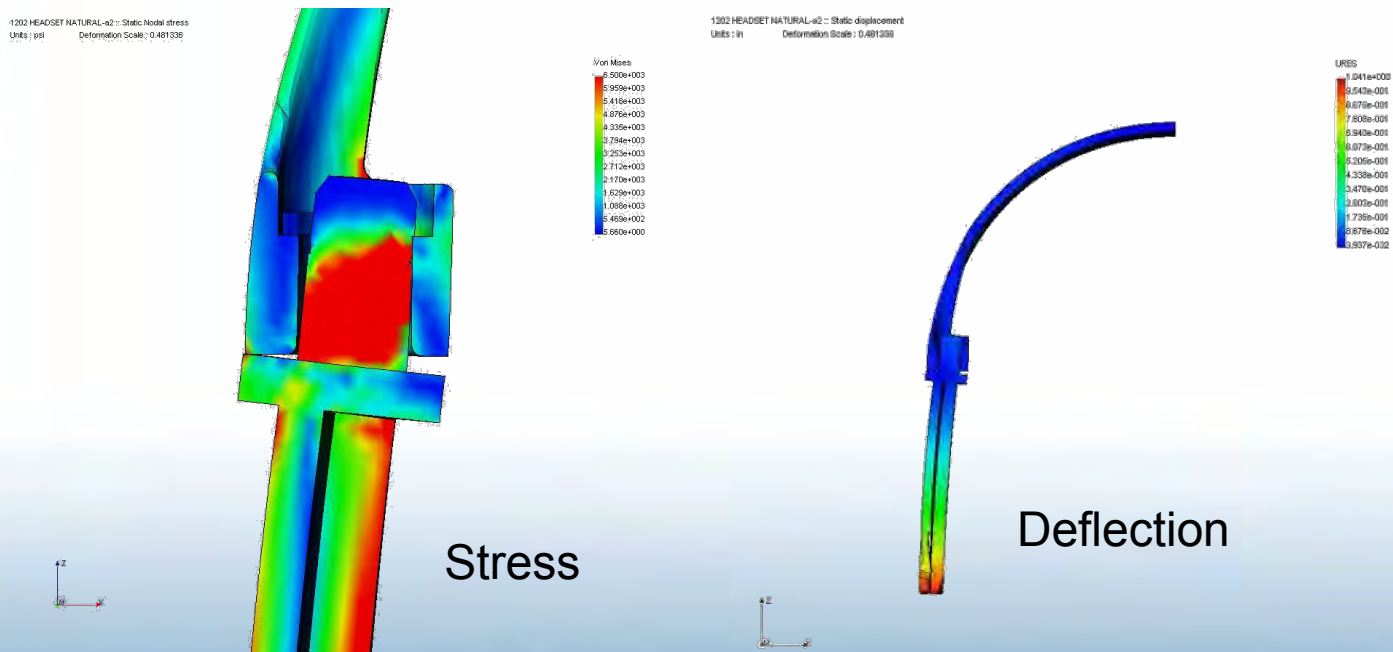
Deflection

Folding Headband behavior

- ## The rapid prototype result
    - ABS simulated urethane material, from Silicone tooling
    - These parts behaved almost perfectly !
    - Possibly too much deflection with a non-linear / plastic material to accurately analyze, or an incorrect assumption made.

High Quality Functional Rapid Prototyping

- **Projet 6000, 50 micron Z step, & 18 micron in X Y:-**
  - Now being used for Bespoke manufacturing.
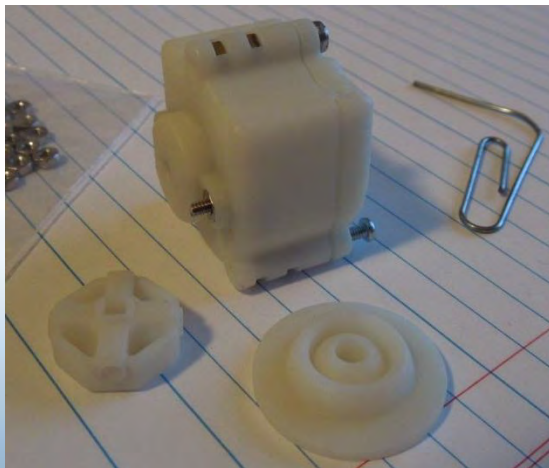
Latest Rapid Prototyping equipment

- **Projet 6000, 50 micron Z step, 18 micron in X Y:-**
  - iPro larger machine but lower resolution at 100 micron.
  - Creates an ABS / Polyprop like material part.
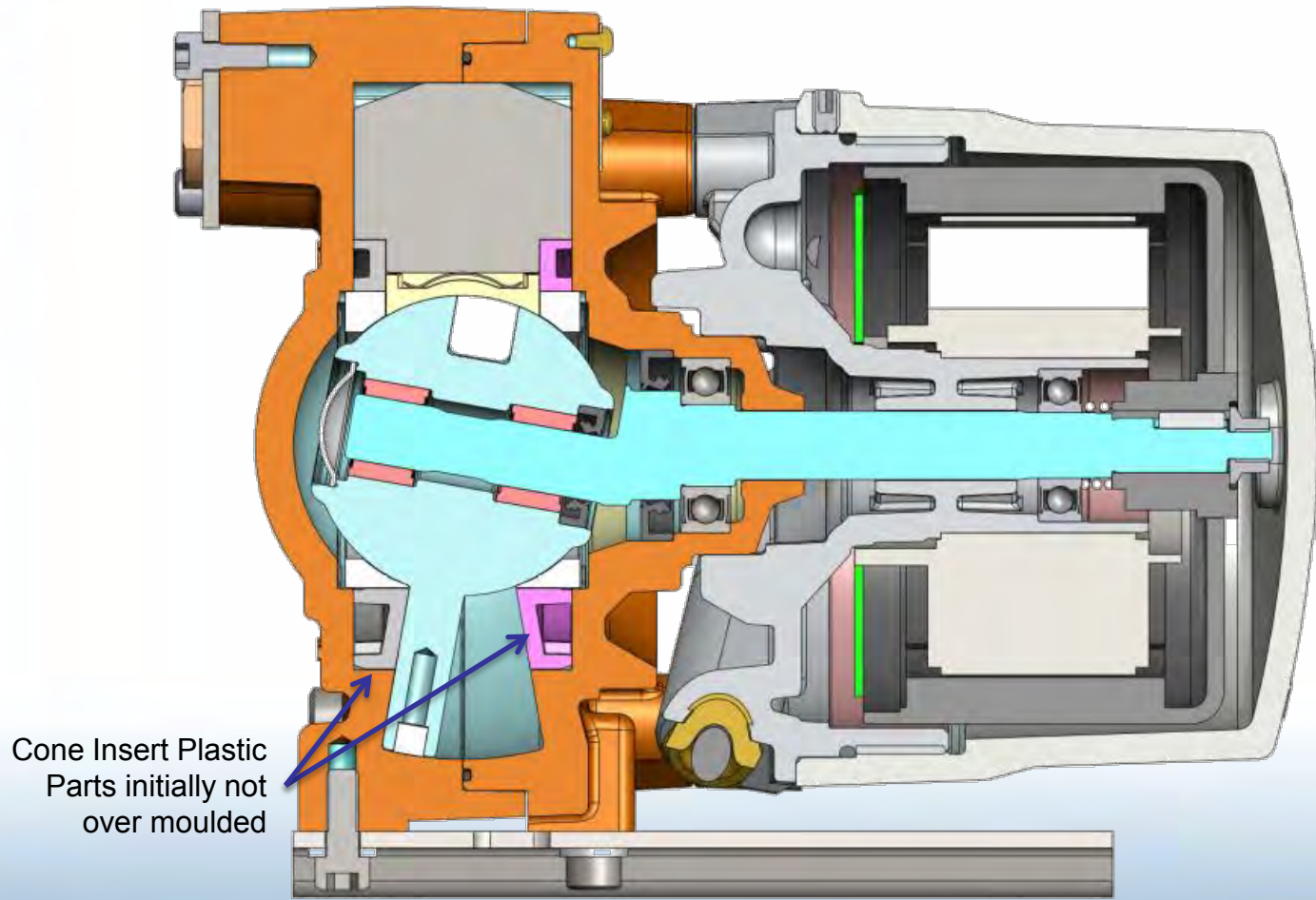
More Functional Prototyping

- **SLS, Nylon, Produced for trade show :-**

SLS still produces Nylon / Strong parts best at a lower resolution, however a little bit furry surface finish wise.
The 6 Units shown here were made for a trade show on this machine on the left.

Warpage of Parts when in Use

- **The Cone Insert from Vapour Pump suffered from this problem:-**

Cone Insert Plastic Parts initially not over moulded

Warpage of Parts when in Use

- **The Cone Insert from Vapour Pump suffered from this problem:-**

Cone Insert Plastic Parts
Press Fitted & Post / Co Machined
with Cast Iron material for accuracy,
initially not over moulded

Elevated operating temperatures of 80deg C
cause a small deflection in the centre of the
Cone Insert that affected performance.
Over Molded Cast Iron substrate to be used
to resolve this.

Overmoulded Plastic Parts

- **Over moulding onto a metal sub-straight, this worked a trick to stabilize Plastic Geometry, both off Tool and in Action:-**
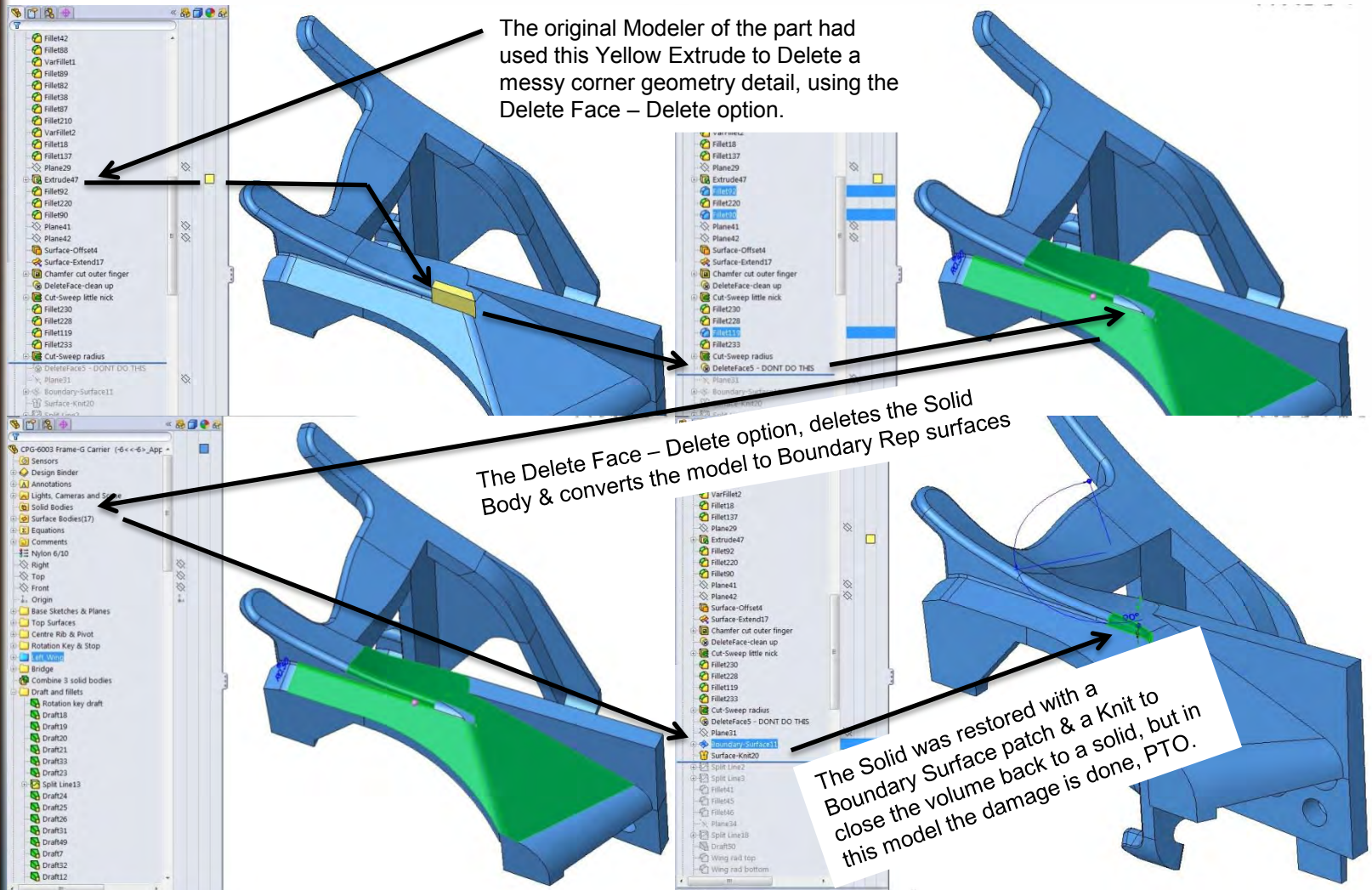
Overmoulded Plastic Parts

- **Over moulded Plastic Spherical Bearing (machined in place for accuracy) this outperformed the Bronze bearing material in life testing by an astonishing margin:-**
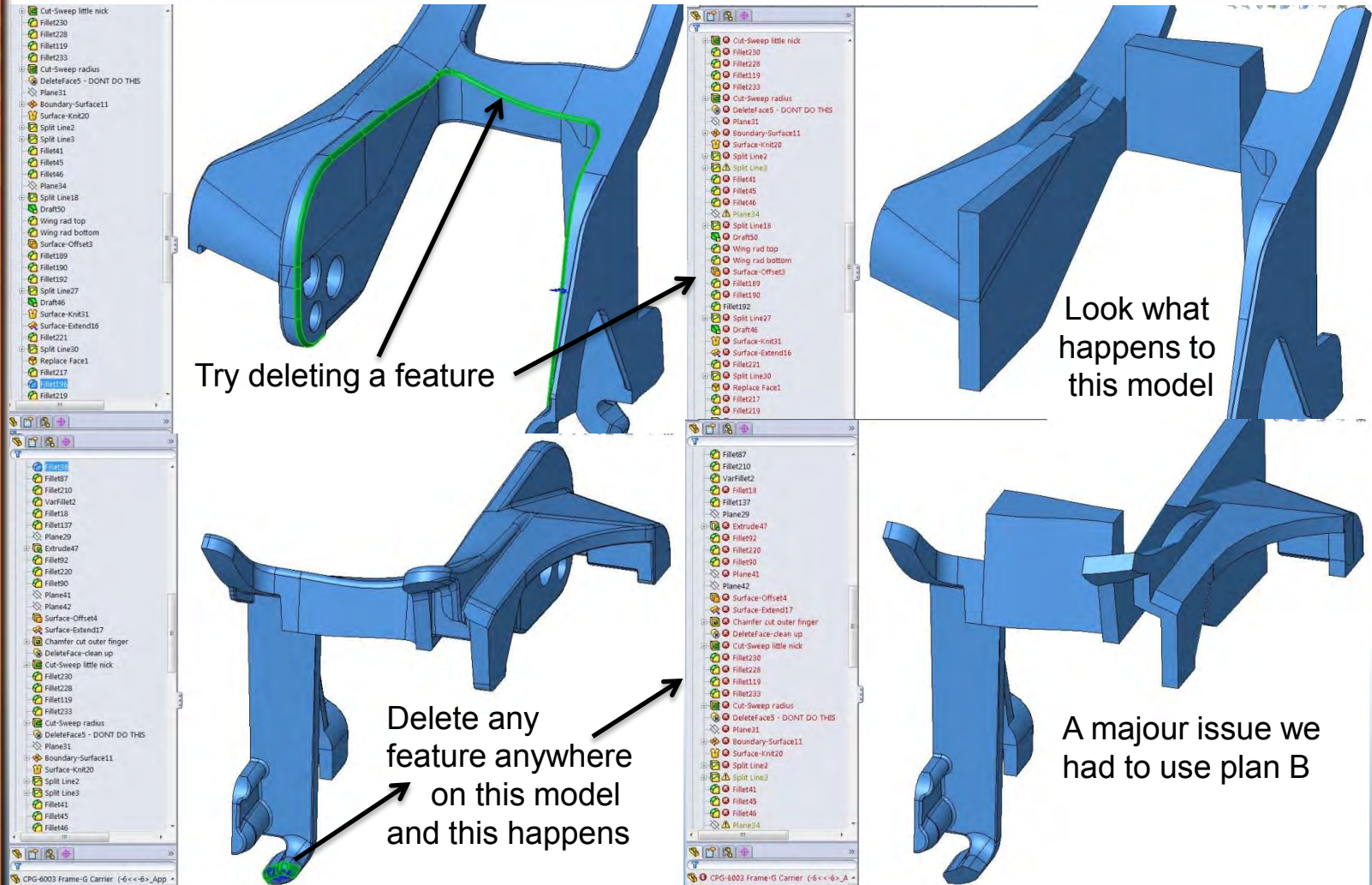
An Overmould Modification job

- **Don't use Delete Face – Delete option in the SolidWorks function unless you are desperate! Why do I say this:-**


The original Modeler of the part had used this Yellow Extrude to Delete a messy corner geometry detail, using the Delete Face – Delete option.

The Delete Face – Delete option, deletes the Solid Body & converts the model to Boundary Rep surfaces

The Solid was restored with a Boundary Surface patch & a Knit to close the volume back to a solid, but in this model the damage is done, PTO.

An Overmould Modification job

- **Don't use Delete Face in SolidWorks function unless you are desperate! Here is what can happen:-**


Try deleting a feature

Look what happens to this model

Delete any feature anywhere on this model and this happens

A majour issue we had to use plan B

The Delete Face Options

- **Suggest that you don't use Delete Face function with the Delete option in SolidWorks, as it destroys the Solid body history!**
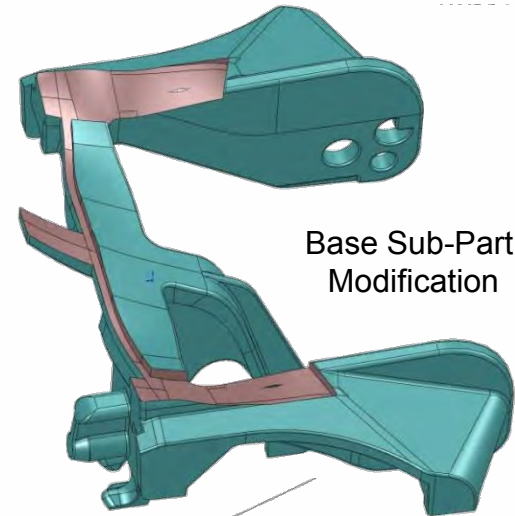

Bad option

Independent Boundary Surface result

Appears not so bad

Fill result
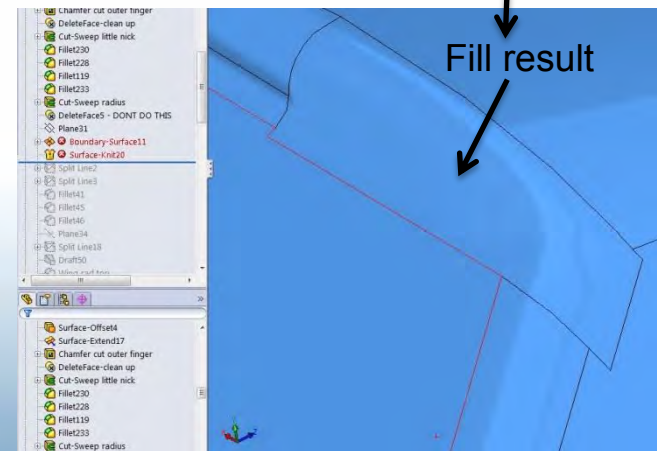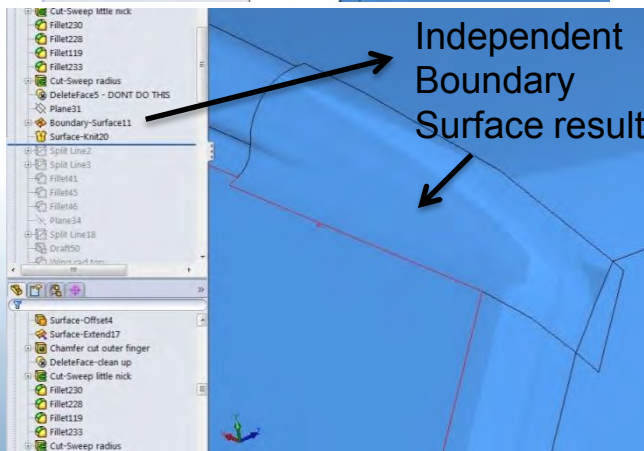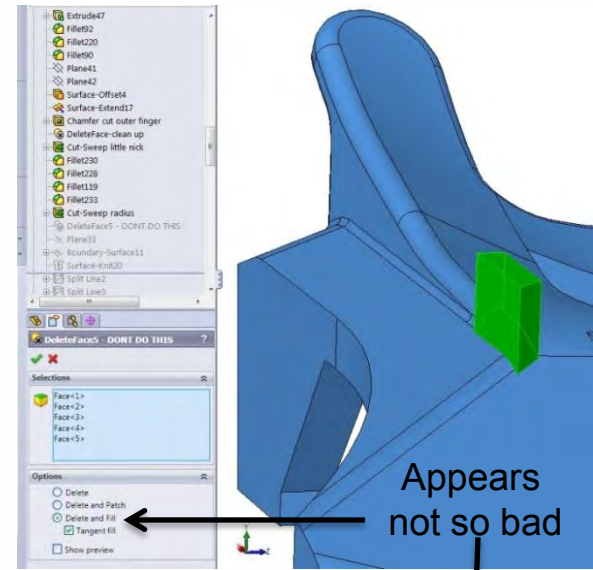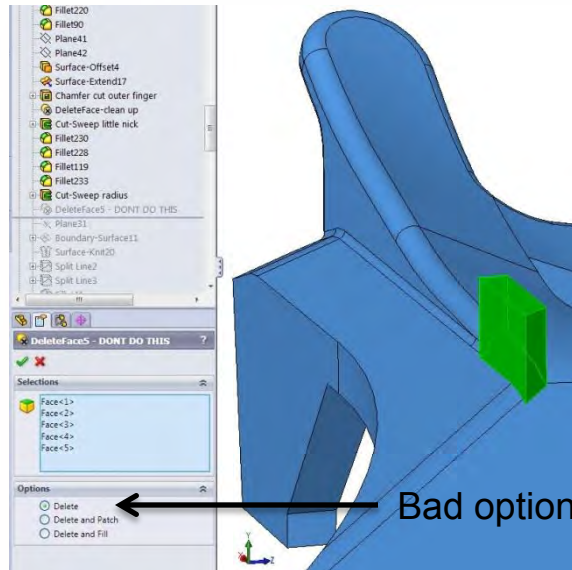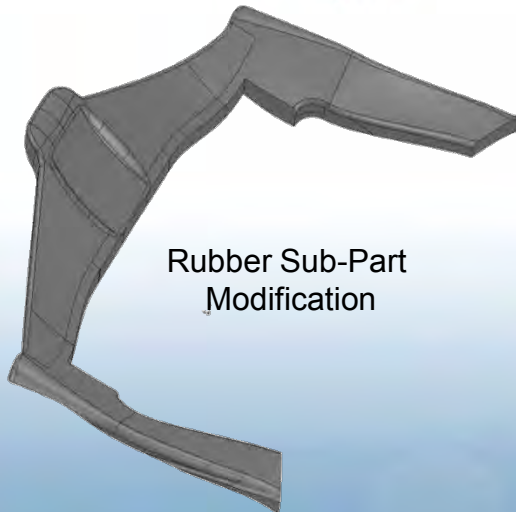
Completed Overmould job

- **That Delete Face Delete option function in SolidWorks made this job really hard work:-**
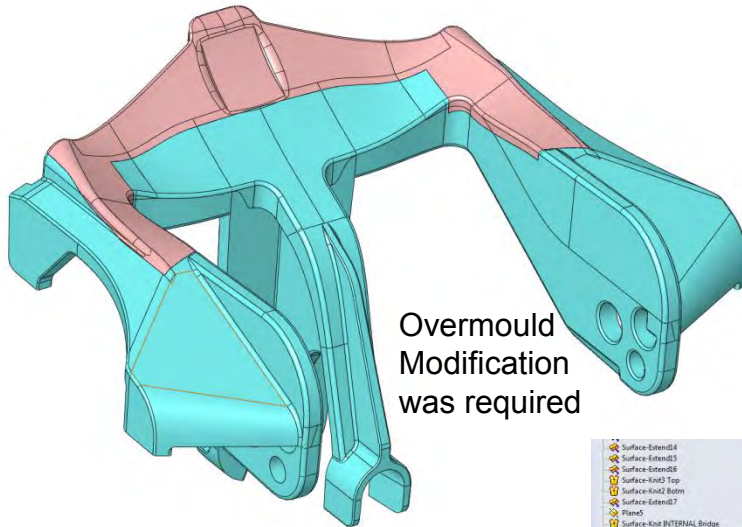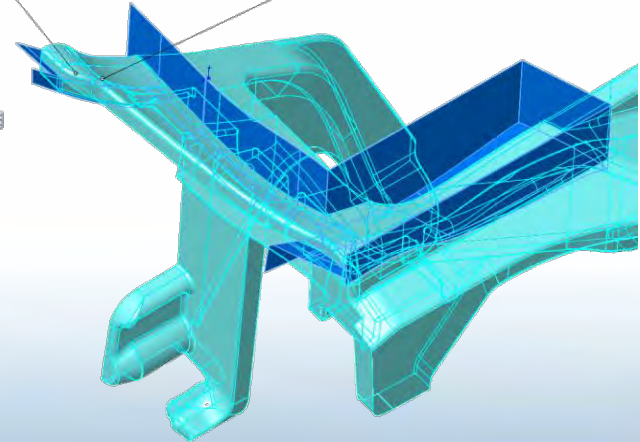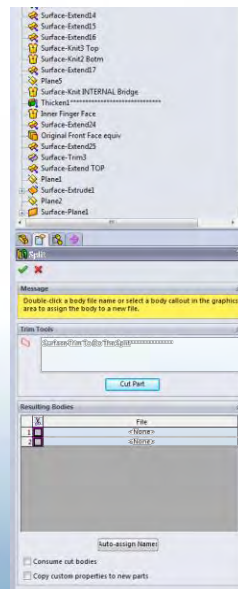
Overmould
Modification
was required

Base Sub-Part
Modification

Rubber Sub-Part
Modification

Major effort to get here, had to export & re-import from a Parasolid to stabilize the base geometry, Then modify the shape & finally split the part.
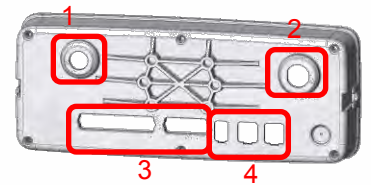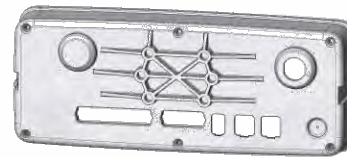
Inserting Tools

- **Using 4 x 2 Independent Inserts on the Rear Cover of this product we produced 6 different Permutations / Models for the client.**

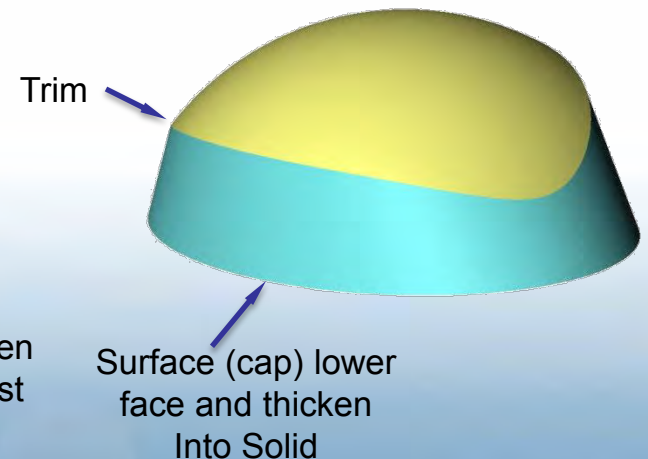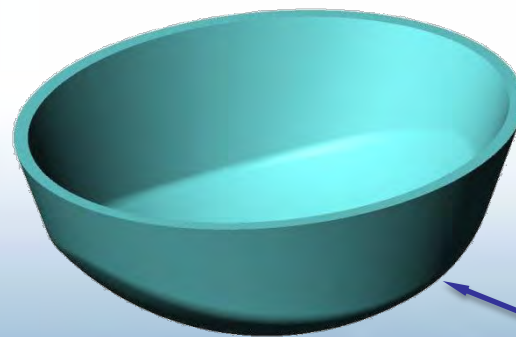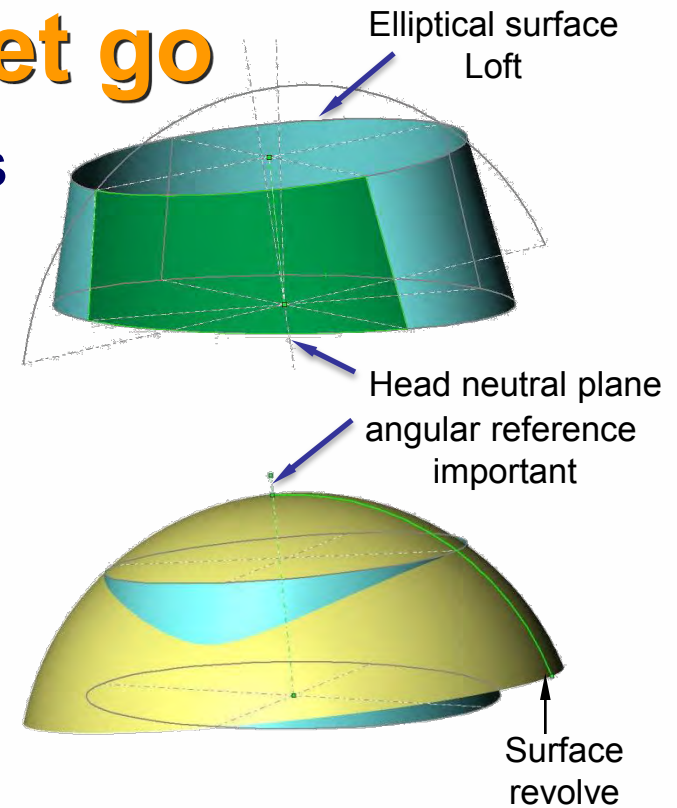PDT 3000 Messaging Device

Off Tool Samples

IP 65 Option

6 Model Permutations from 4 Inserted Areas

Surfaces from the get go

- ## Cup constructed with Surfaces

  - ### Elliptical Surface Loft
    - Depth & angle references 3D ear neutral plane from assembly

  - ### Surface Revolve & Trim
    - Important for the functionally of spherical bearing surface

  - ### Surface Cap & Thicken

  - ### Fillet & Shell

Elliptical surface Loft

Head neutral plane angular reference important

Surface revolve

Trim

Fillet then Shell last

Surface (cap) lower face and thicken Into Solid

Imbedded Surfaces

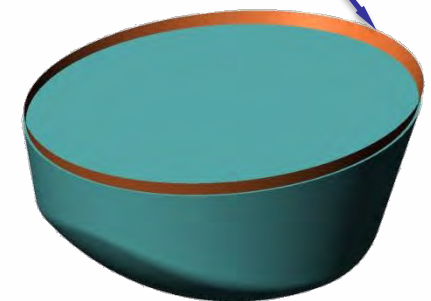- ## Cup Internal Geometry
  - The acoustic enclosure
    - Rim required for Driver mount at an ear-friendly angle
  - Strategic surfaces created
    - Surface offsets created at points in the feature tree i.e. Just before & after the shell operation
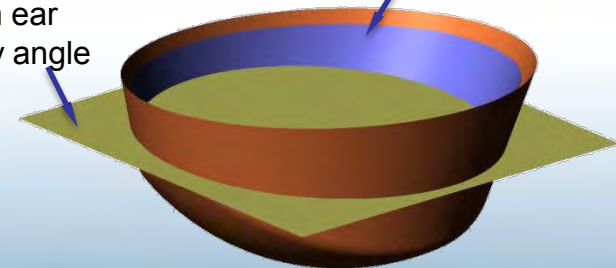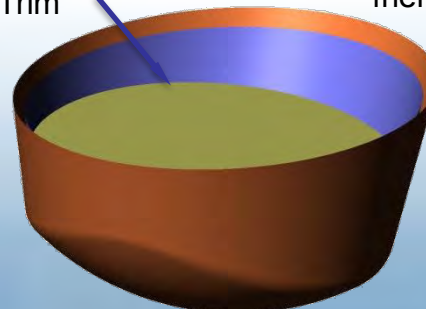   - These are used to help create the future geometry
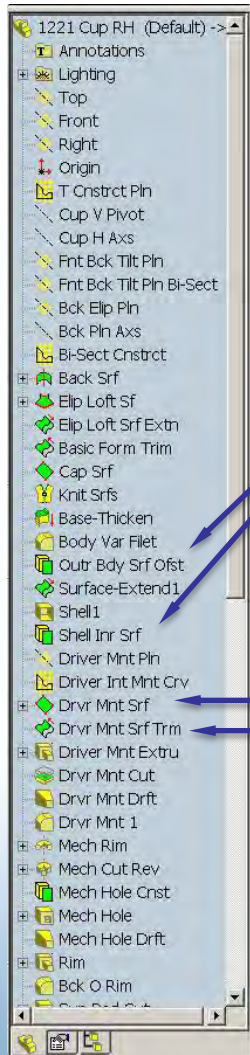
Complete Cup

Outr Bdy Srf Ofst (brown) Before Shell

Shel Inr Srf (blue) after Shell, Solid body hidden

Drvr Mnt Srf (yellow) @ an ear friendly angle

Drvr Mnt & Shell Inr Srf Trim

Feature tree:

1221 Cup RH (Default) ->
- Annotations
- Lighting
- Top
- Front
- Right
- Origin
- T Cnstrct Pln
- Cup V Pivot
- Cup H Axs
- Fnt Bck Tilt Pln
- Fnt Bck Tilt Pln Bi-Sect
- Bck Elip Pln
- Bck Pln Axs
- Bi-Sect Cnstrct
- Back Srf
- Elip Loft Sf
- Elip Loft Srf Extn
- Basic Form Trim
- Cap Srf
- Knit Srfs
- Base-Thicken
- Body Var Filet
- Outr Bdy Srf Ofst
- Surface-Extend1
- Shell1
- Shell Inr Srf
- Driver Mnt Pln
- Driver Int Mnt Crv
- Drvr Mnt Srf
- Drvr Mnt Srf Trm
- Driver Mnt Extru
- Drvr Mnt Cut
- Drvr Mnt Drft
- Drvr Mnt 1
- Mech Rim
- Mech Cut Rev
- Mech Hole Cnst
- Mech Hole
- Mech Hole Drft
- Rim
- Bck O Rim
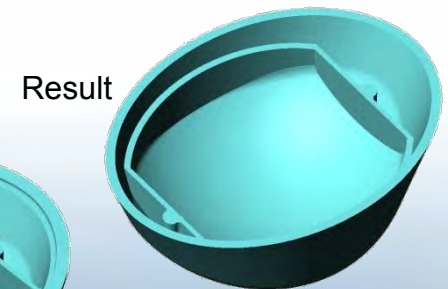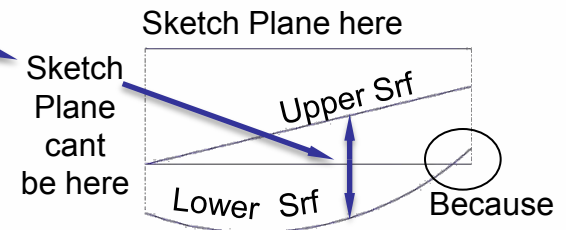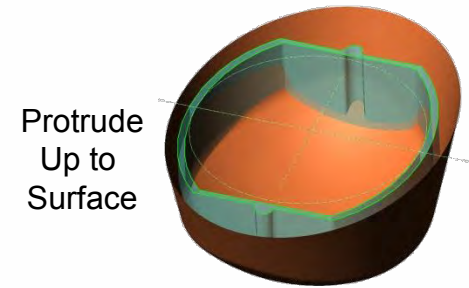
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Completing the Cup

- ## Why these surfaces?

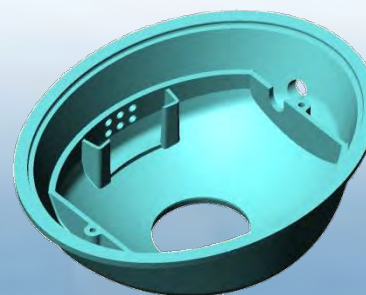 - ### Create Solids up to surface

    - Enables SolidWorks to easily build the required geometry up to surface as it is offset into the existing part wall by 0.5mm

    - **Note:-** The current up to surface functionality has been improved since this model was created, but you still can't have overlaps like this shown here

  - ### Other Surface used for Cut

    - The top of the Driver housing mounting rim is then cut away with the other strategically placed surface trim in the feature tree.
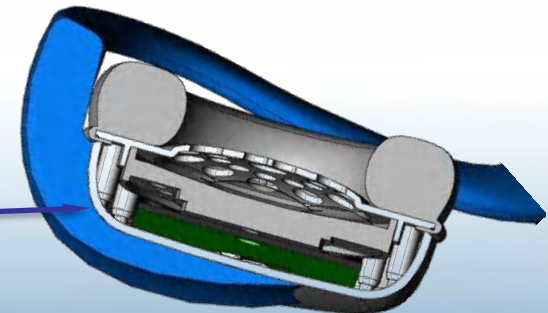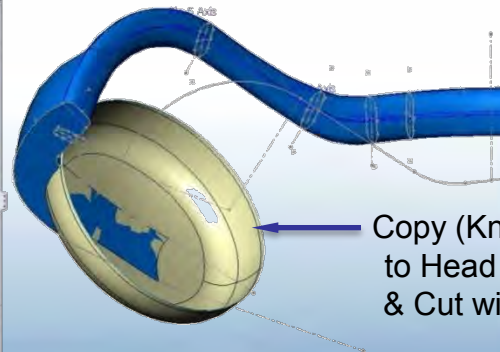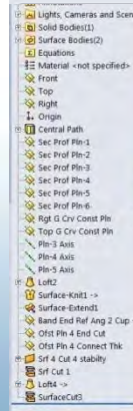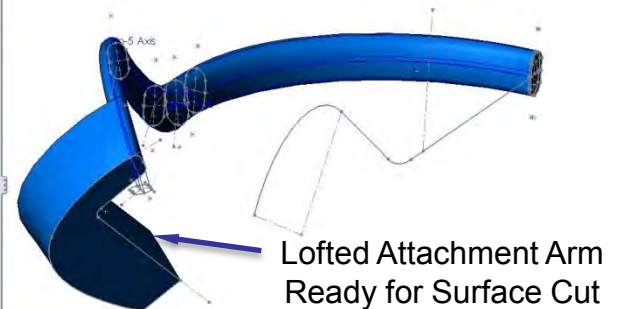
Protrude Up to Surface

Cut with Surface

Sketch Plane here

Sketch Plane cant be here

Upper Srf

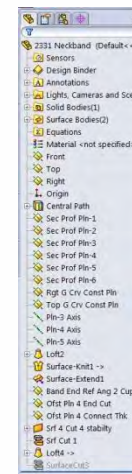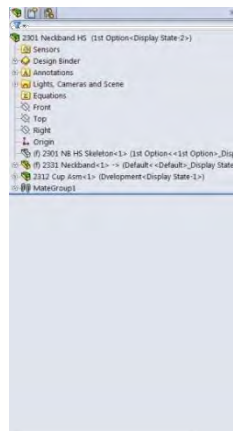Lower Srf

Because

Result
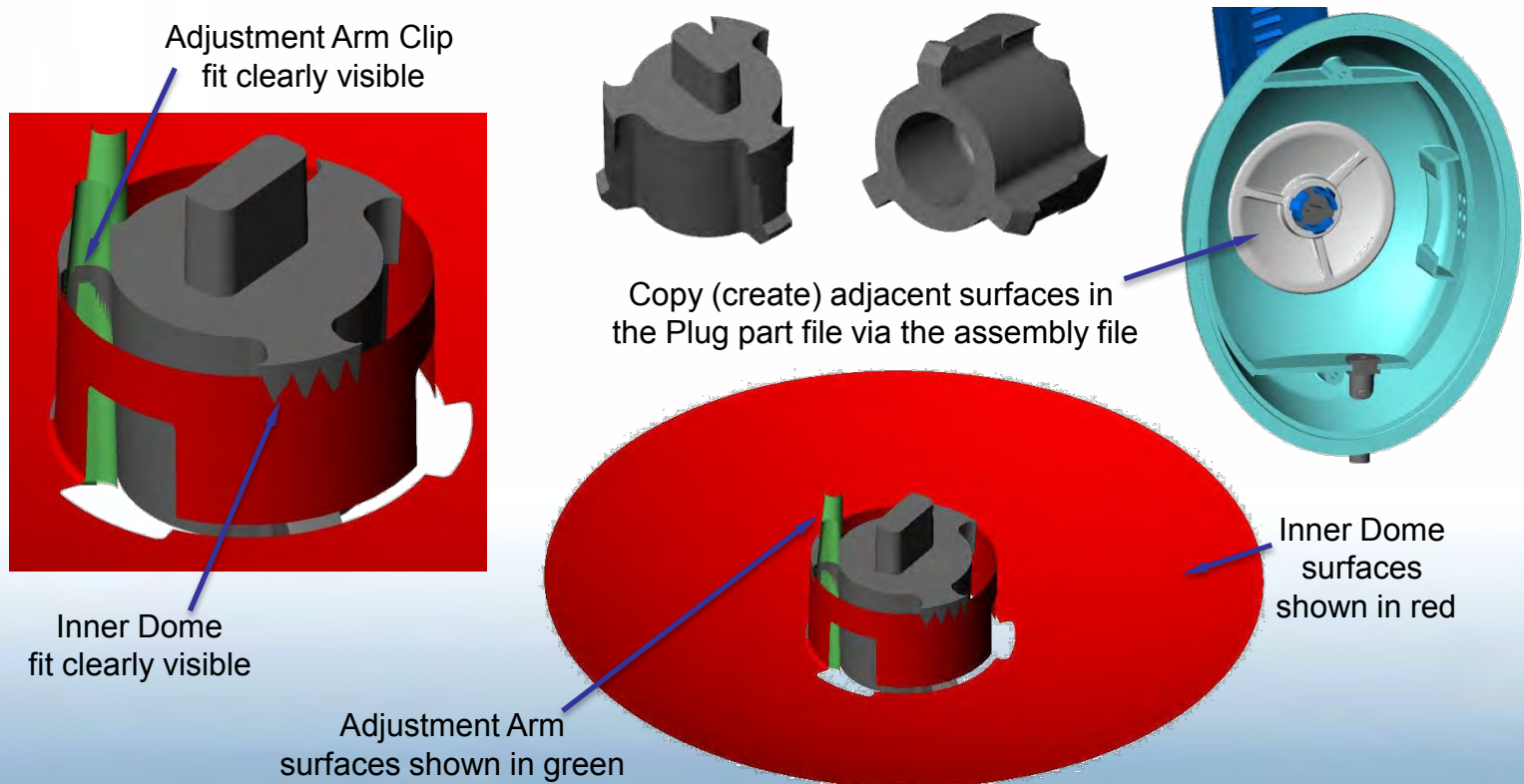
Complete Cup

Copying Surfaces across Parts

- ## Neckband Headset, half shown.

  - Tricky 3D geometry / Part Interface relationship(s).
  - Copy surface was used from inside the assembly across parts, this ensured good part fit-up.

Modify Part in Asm & Surface Knit off Back Face of Cup

Lofted Attachment Arm Ready for Surface Cut

Copy (Knit) Surface to Head Band Part & Cut with Surface

More Copy Surface

- **Small rubber plug created purely from surfaces**
  - A good Acoustic seal, snug fit was required.
  - Copy surface was used from the adjacent assembly parts, to ensure fit, volume closed (surface knitted) to form the Solid.

Adjustment Arm Clip fit clearly visible

Copy (create) adjacent surfaces in the Plug part file via the assembly file

Inner Dome fit clearly visible

Inner Dome surfaces shown in red

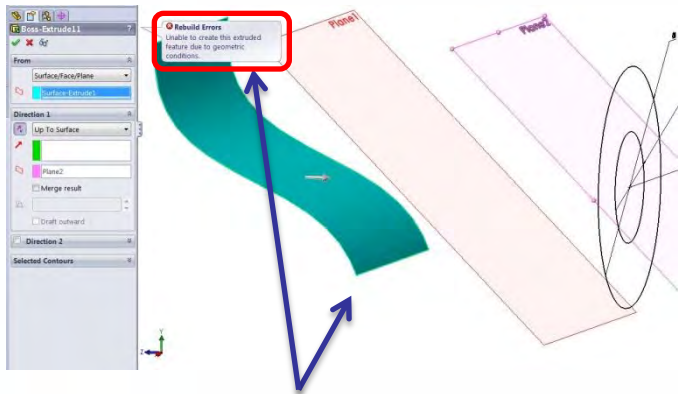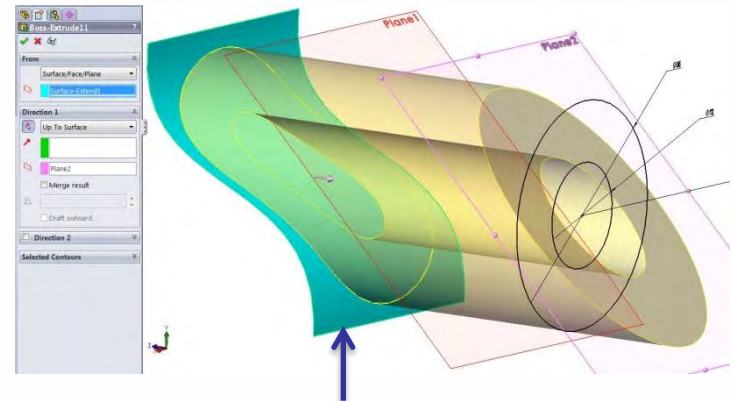Adjustment Arm surfaces shown in green

Upto Surface & some other pointers

- **Extrude or cut up to surface**
  - The Surface must be big enough for the projection to hit the Surface, use the Surface Extend & Trim function if needed to achieve this

Surface not long enough

Add a Surface Extend

- **A couple of other points to mention**
  - Preserve sketch geometry toggle sketch entities to use for Construction
  - Backup your model often, I use Ctrl C & Ctrl V in explorer. I haven't relied on auto recover, & this is really easy to do.
  - Finally we have full Conic support in the SWX 2013 sketcher with an rho value so that a hyperbola can be created, = constant rate of change of curvature within the curve. I might even upgrade for that!

Practical Plastic Part Design

Lindsay Dalziel
Design Engineer

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