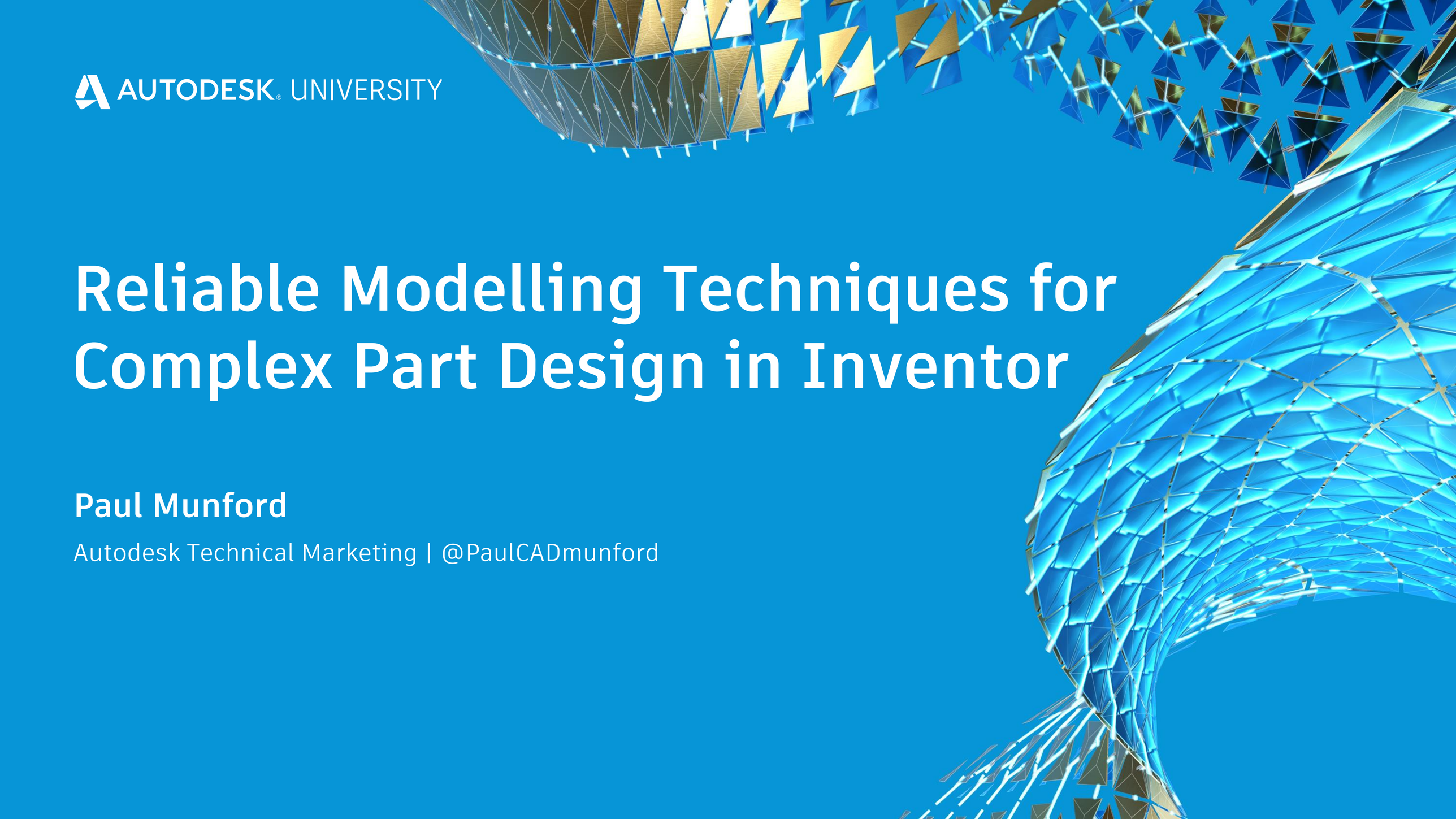
AUTODESK. UNIVERSITY

Reliable Modelling Techniques for Complex Part Design in Inventor

Paul Munford

Autodesk Technical Marketing | @PaulCADmunford



Paul Munford

@PaulCADmunford

Carpenter

Drafter (AutoCAD & Inventor)

CAD/CAM Manager

Trainer & Consultant

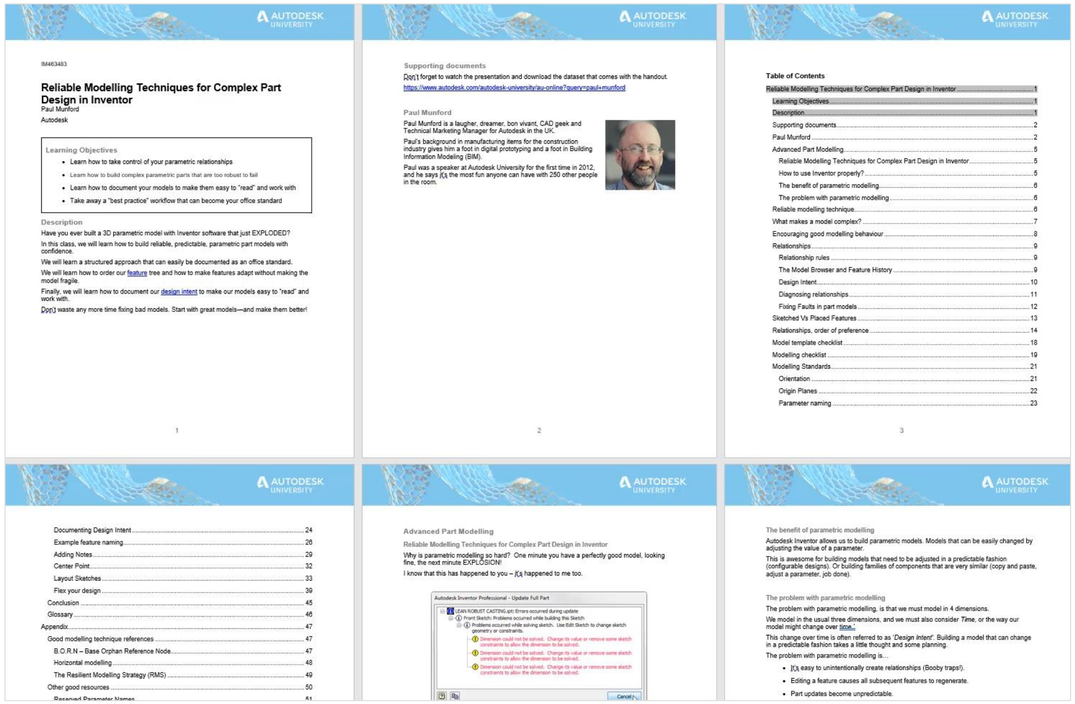
Technical Marketing Manager for Autodesk

Reliable Modelling Techniques for Complex Part Design in Inventor

Learning objectives

- Learn how to take control of your parametric relationships
- Learn how to build complex parametric parts that are too robust to fail
- Learn how to document your models to make them easy to “read” and work with
- Take away a “best practice” workflow that can become your office standard

Downloads



IM463483

Reliable Modelling Techniques for Complex Part Design in Inventor

Paul Munford

Autodesk

Learning Objectives

- Learn how to take control of your parametric relationships
- Learn how to build complex parametric parts that are too robust to fail
- Learn how to document your models to make them easy to "read" and work with
- Take away a "best practice" workflow that can become your office standard

Description

Have you ever built a 3D parametric model with Inventor software that just EXPLODED?

In this class, we will learn how to build reliable, predictable, parametric part models with confidence.

We will learn a structured approach that can easily be documented as an office standard.

We will learn how to order our feature tree and how to make features adapt without making the model fragile.

Finally, we will learn how to document our design intent to make our models easy to "read" and work with.

Don't waste any more time fixing bad models. Start with great models—and make them better!

1

Supporting documents

Don't forget to watch the presentation and download the dataset that comes with the handout.

https://www.autodesk.com/autodesk-university/au-online?query=paul+munford

Paul Munford

Paul Munford is a laugher, dreamer, bon vivant, CAD geek and Technical Marketing Manager for Autodesk in the UK.

Paul's background in manufacturing items for the construction industry gives him a foot in digital prototyping and a foot in Building Information Modeling (BIM).

Paul was a speaker at Autodesk University for the first time in 2012, and he says it's the most fun anyone can have with 250 other people in the room.

2

Table of Contents

3

Advanced Part Modelling

Reliable Modelling Techniques for Complex Part Design in Inventor

Why is parametric modelling so hard? One minute you have a perfectly good model, looking fine, the next minute EXPLOSION!

I know that this has happened to you – it's happened to me too.

The benefit of parametric modelling

Autodesk Inventor allows us to build parametric models. Models that can be easily changed by adjusting the value of a parameter.

This is awesome for building models that need to be adjusted in a predictable fashion (configurable designs). Or building families of components that are very similar (copy and paste, adjust a parameter, job done).

The problem with parametric modelling

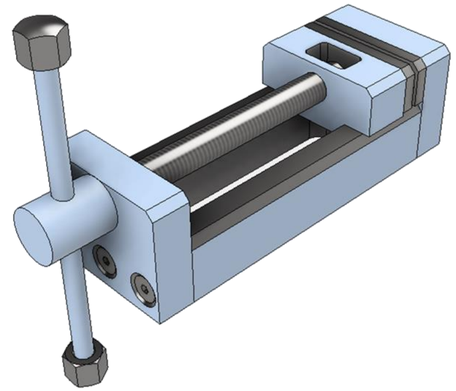
The problem with parametric modelling, is that we must model in 4 dimensions.

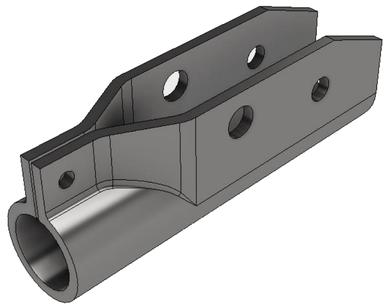
We model in the usual three dimensions, and we must also consider *Time*, or the way our model might change over time.

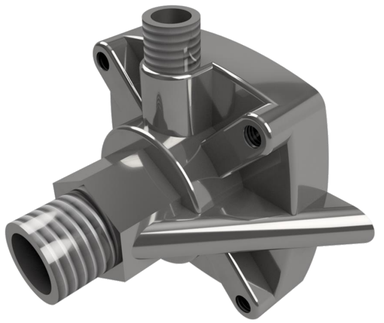
This change over time is often referred to as *'Design Intent'*. Building a model that can change in a predictable fashion takes a little thought and some planning.

The problem with parametric modelling is ...

- It's easy to unintentionally create relationships (Booby traps!).
- Editing a feature causes all subsequent features to regenerate.
- Part updates become unpredictable.



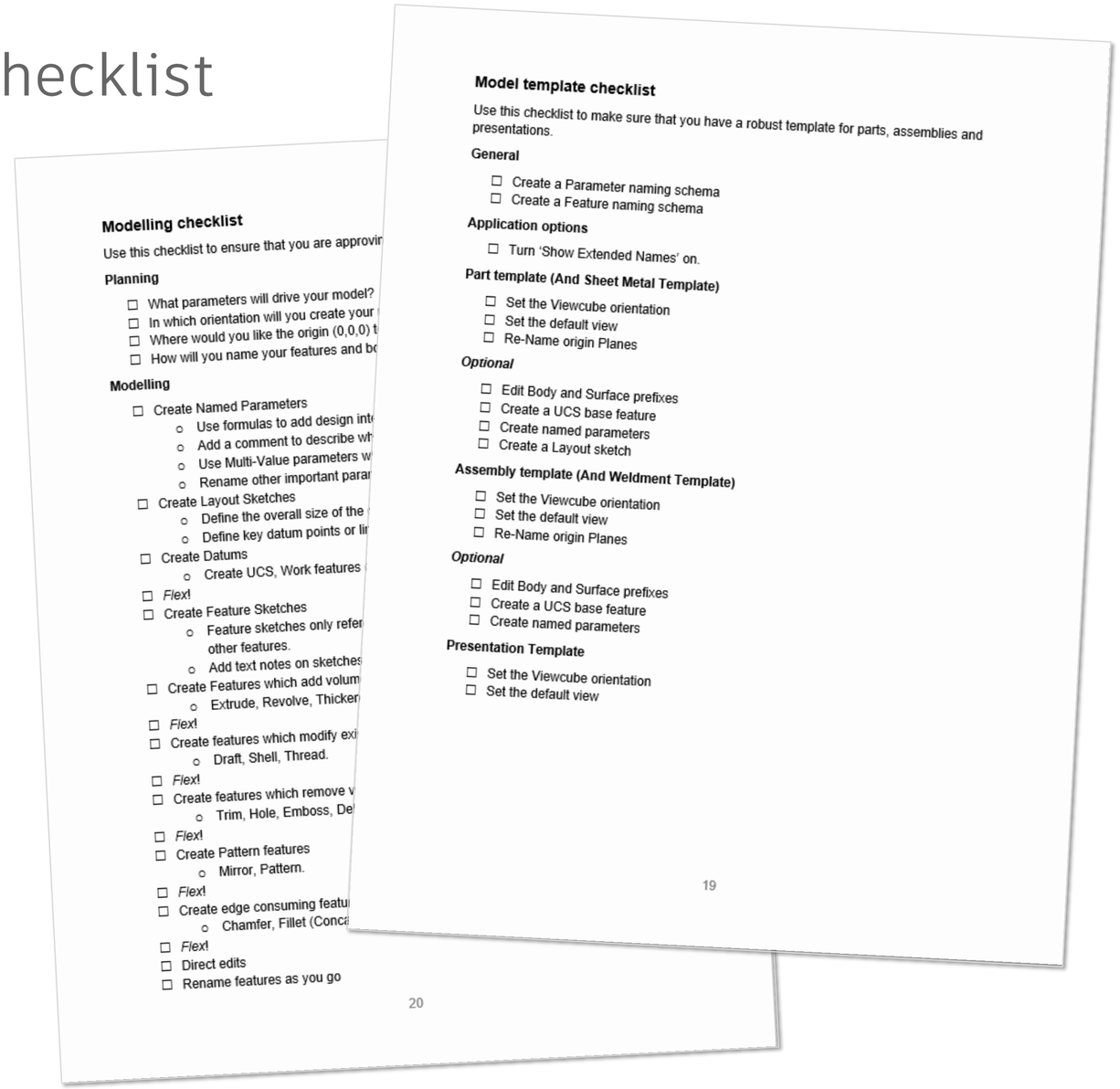




- Download the handout and dataset from the class page:
- Or use this link: **http://cadso.co/IM463483_DOWNLOAD**

Checklists

- Template/Application options checklist
- Modelling Checklist



Model template checklist

Use this checklist to make sure that you have a robust template for parts, assemblies and presentations.

General

- ☐ Create a Parameter naming schema
- ☐ Create a Feature naming schema

Application options

- ☐ Turn 'Show Extended Names' on.

Part template (And Sheet Metal Template)

- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes

Optional

- ☐ Edit Body and Surface prefixes
- ☐ Create a UCS base feature
- ☐ Create named parameters
- ☐ Create a Layout sketch

Assembly template (And Weldment Template)

- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes

Optional

- ☐ Edit Body and Surface prefixes
- ☐ Create a UCS base feature
- ☐ Create named parameters

Presentation Template

- ☐ Set the Viewcube orientation
- ☐ Set the default view

19

Modelling checklist

Use this checklist to ensure that you are approvi

Planning

- ☐ What parameters will drive your model?
- ☐ In which orientation will you create your
- ☐ Where would you like the origin (0,0,0) t
- ☐ How will you name your features and bo

Modelling

- ☐ Create Named Parameters
  - ○ Use formulas to add design int
  - ○ Add a comment to describe wh
  - ○ Use Multi-Value parameters w
  - ○ Rename other important para
- ☐ Create Layout Sketches
  - ○ Define the overall size of the
  - ○ Define key datum points or li
- ☐ Create Datums
  - ○ Create UCS, Work features
- ☐ *Flex*!
- ☐ Create Feature Sketches
  - ○ Feature sketches only refer other features.
  - ○ Add text notes on sketches
- ☐ Create Features which add volum
  - ○ Extrude, Revolve, Thicken
- ☐ *Flex*!
- ☐ Create features which modify exi
  - ○ Draft, Shell, Thread.
- ☐ *Flex*!
- ☐ Create features which remove v
  - ○ Trim, Hole, Emboss, De
- ☐ *Flex*!
- ☐ Create Pattern features
  - ○ Mirror, Pattern.
- ☐ *Flex*!
- ☐ Create edge consuming featu
  - ○ Chamfer, Fillet (Conca
- ☐ *Flex*!
- ☐ Direct edits
- ☐ Rename features as you go

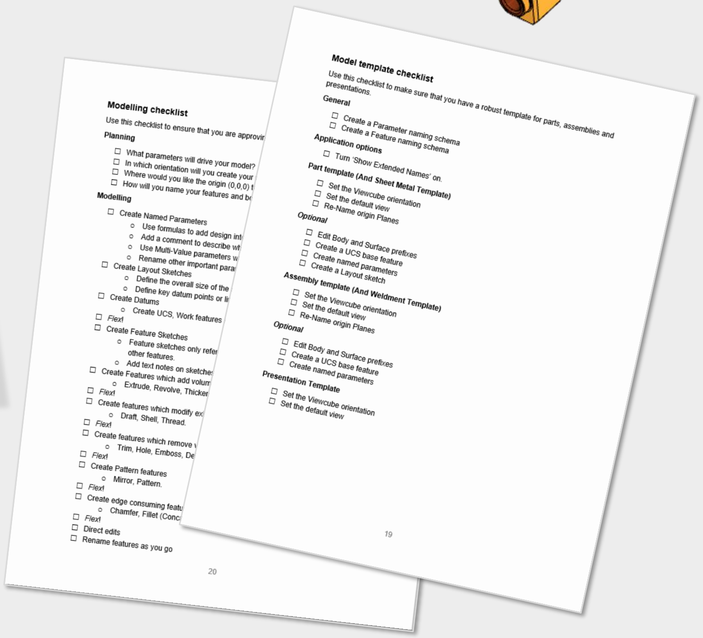
20

AUTODESK. UNIVERSITY

Reliable Techniques for complex Assembly design in Autodesk Inventor

Paul Munford

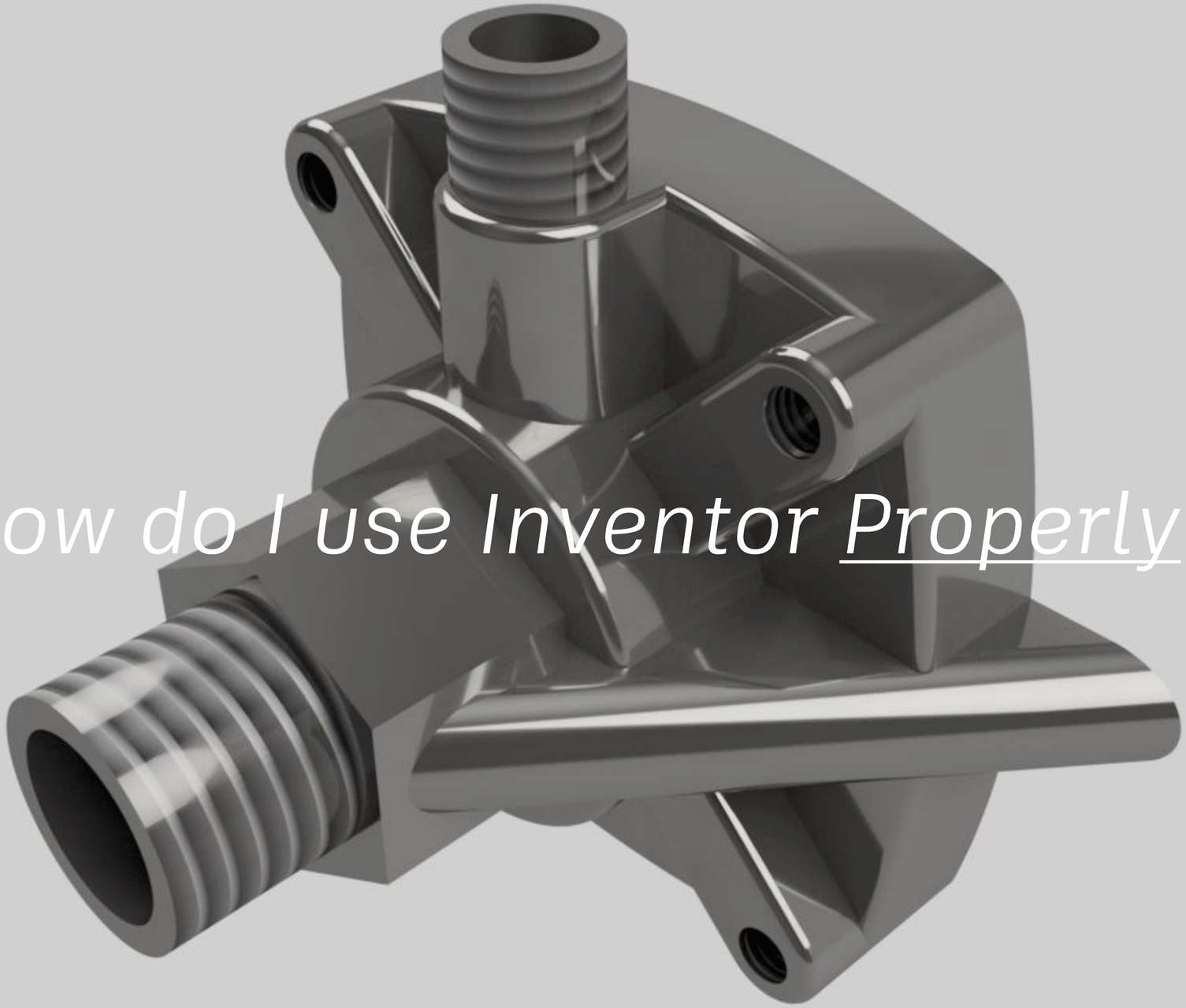
Autodesk Technical Marketing | @PaulCADmunford



http://cadso.co/PaulCADMunford-AU-online

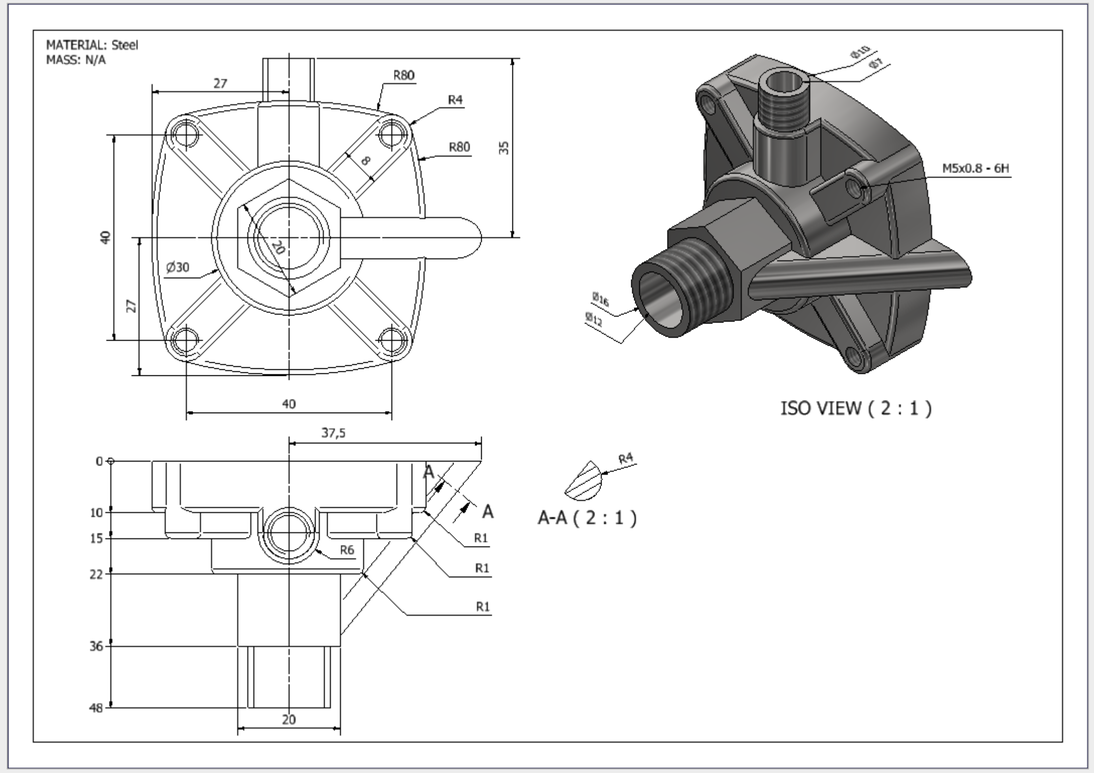
Q&A


Comment (1)
Comments
PM
Please feel free to post your questions in the comments!
We have a be nice policy
Please be positive and constructive
POST

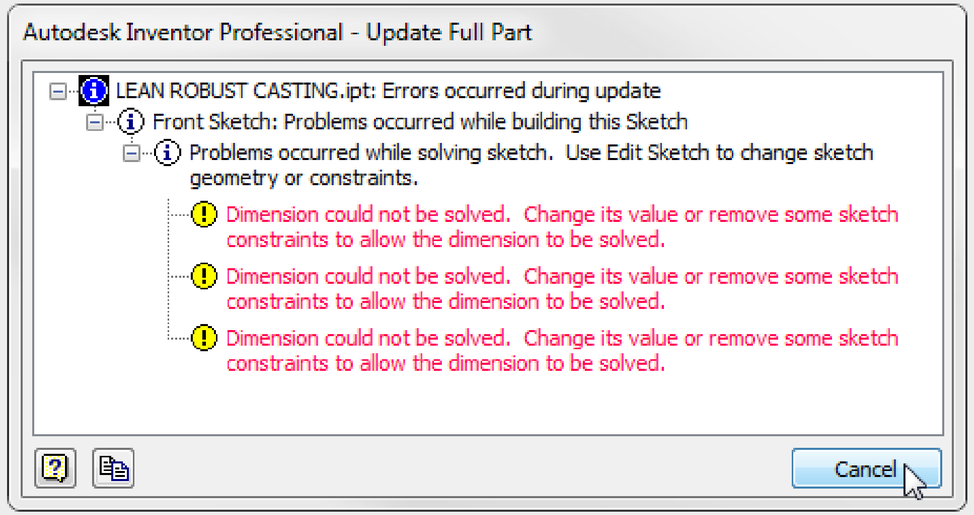


How do I use Inventor Properly?

✓ Correct geometry



✓ Easy to update



iLOGIC WHEEL CONFIGURATOR

WHEEL OPTIONS

WHEEL SIZE	22
WHEEL FINISH	Chrome - Polished
NO OF SPOKES	6
WHEEL PRICE	***$310***

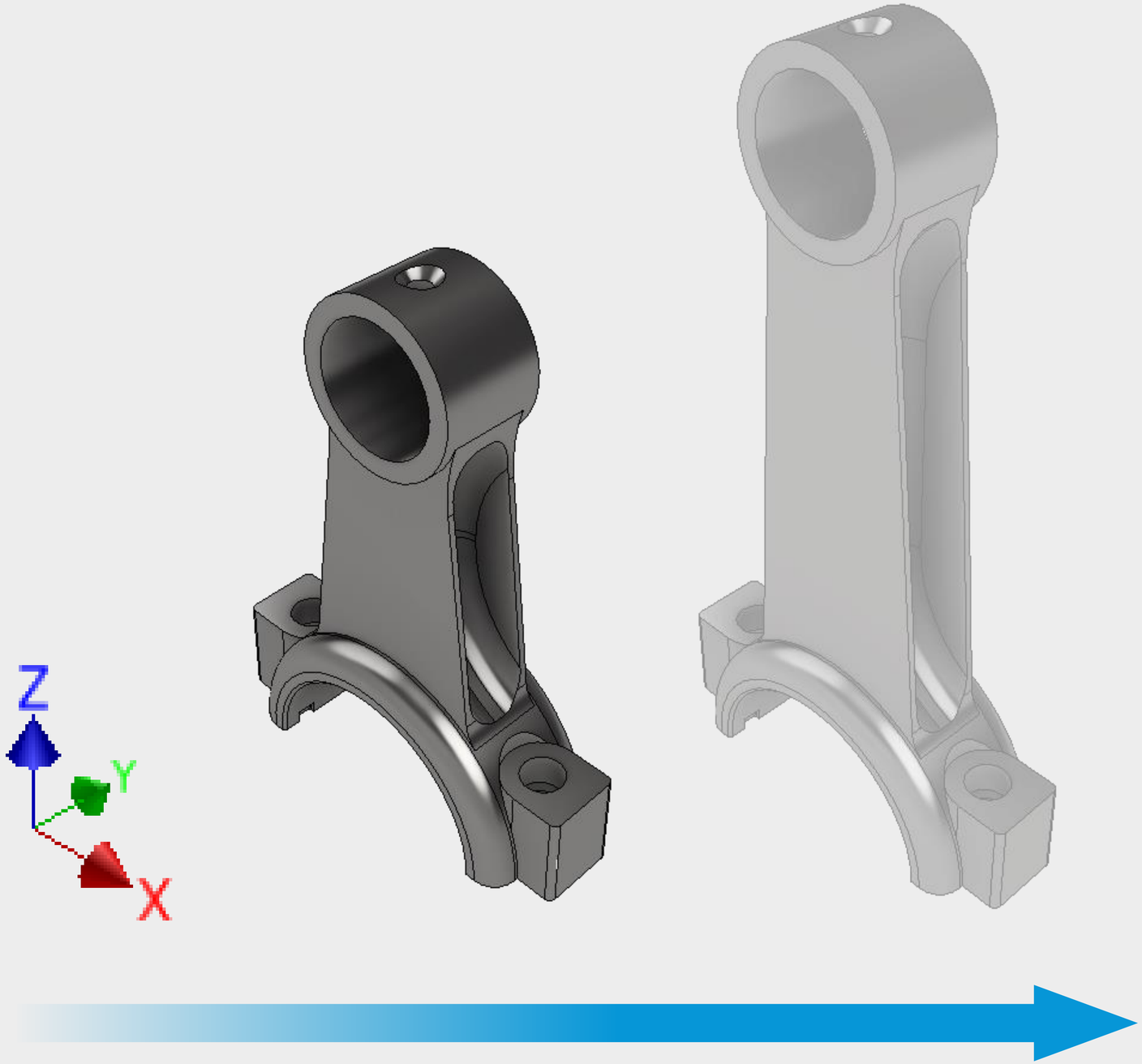
BRAKE OPTIONS

BRAKE MATERIAL	Cast Iron
CALIPER FINISH	Smooth - Light Orange
BRAKE PRICE	***$700***

TOTAL

TOTAL PRICE	***$1010***

Done

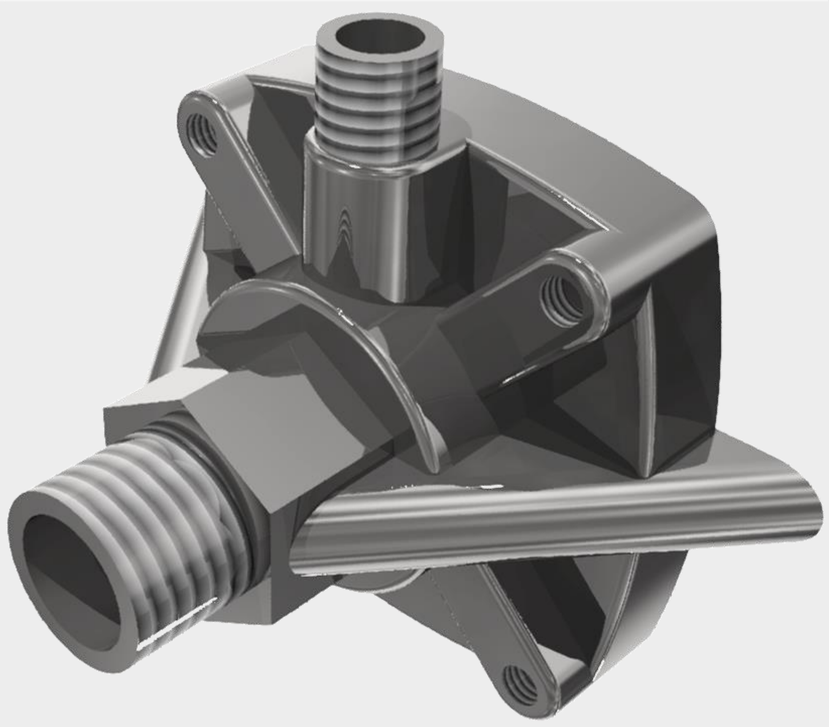

Z
Y
X

Unintentional relationships + Feature **Regeneration**

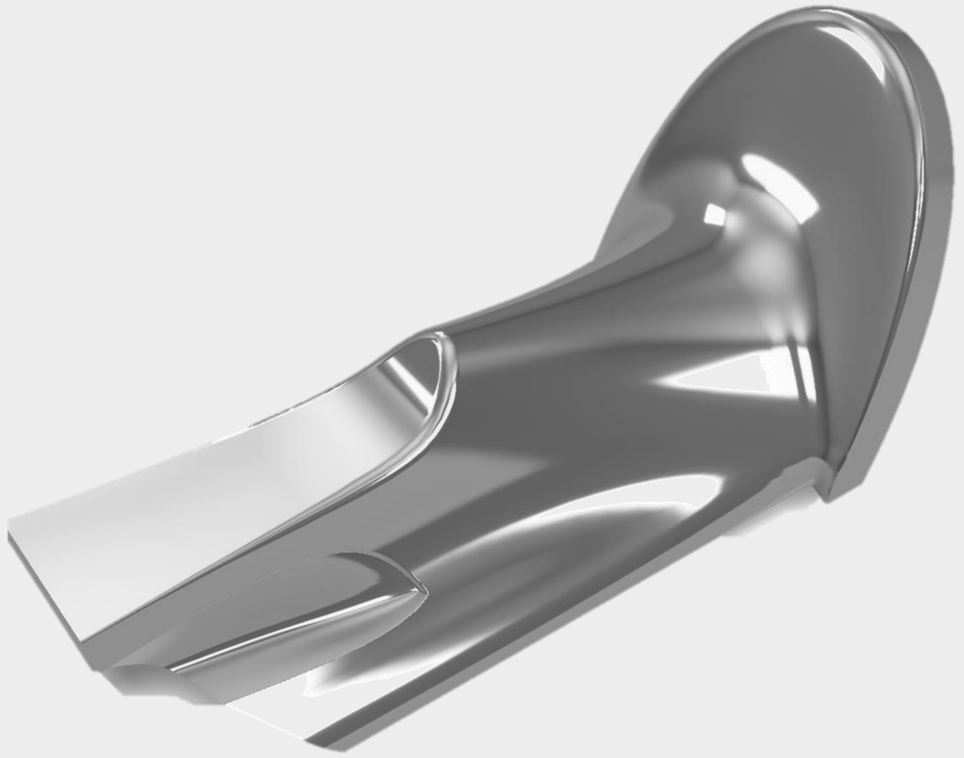
- **Unpredictable** Updates
  - Design intent **lost**
    - Time lost ‘**fixing**’ parts
      - Re-build rather than **Re-use**

Editable models	Design intent is captured
Obvious Models	Design Intent is documented
Reusable Models	Re-use rather than Re-Build

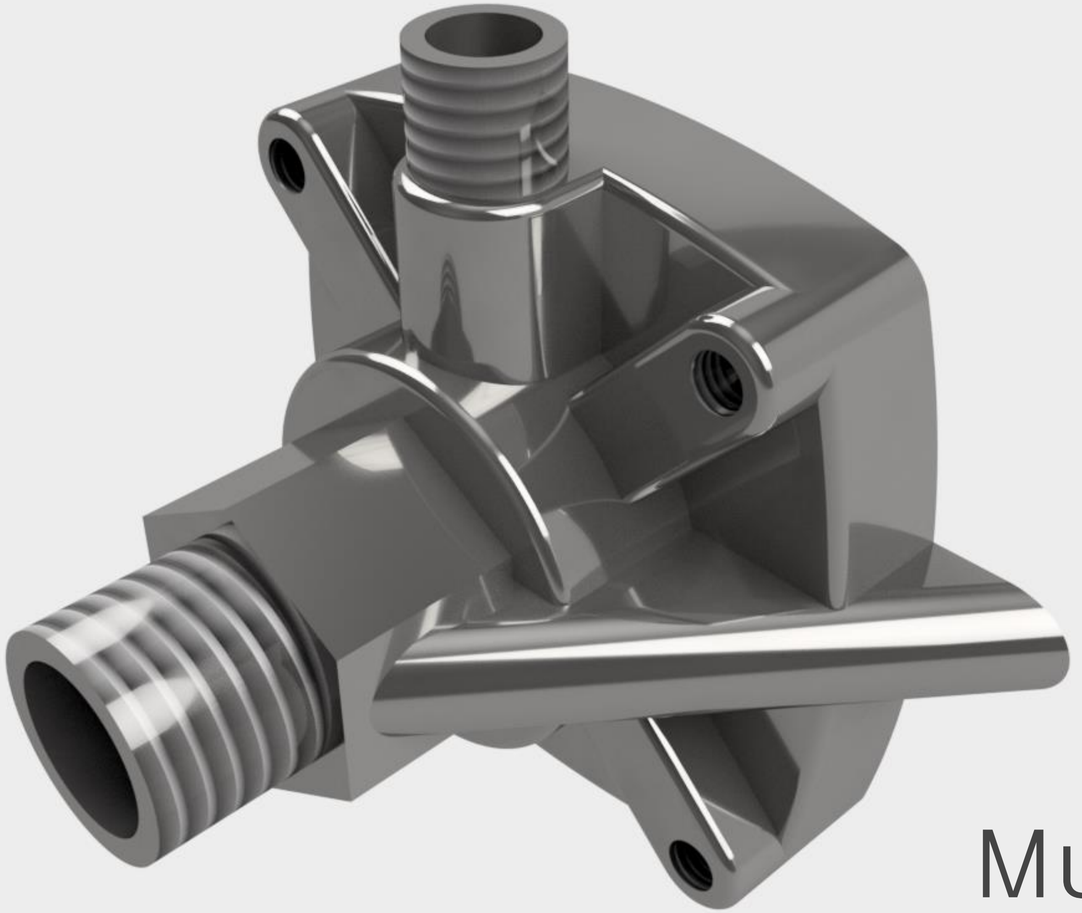
Complex
‘Many Elements’



Complicated
‘Intricate’



- CASTING.ipt
  - Solid Bodies(1)
  - View: Master
  - Origin
  - LAYOUT
  - Layout Sketch
  - Front Sketch
  - Top Sketch
  - Boss Hole Sketch
  - Flange Sketch
  - Top Connector Sketch
  - ADD
  - Base Ext (New Solid x BaseDepth)
  - Boss Hole Ext (Join x BaseDepth + BossHoleDepth)
  - Hex Ext (Join x d21)
  - Flange Ext (Join x FlangeThick)
  - Top Connector Ext 01 (Join To)
  - Top Connector Ext 02 (Join x 35 mm)
  - Boss Center Rev (Join Full)
  - Connector Front Rev (Join Full)
  - REMOVE
  - Hole1 (M5x0.8 x 3 mm ThreadDepth x 5 mm Depth)
  - MODIFY
  - Connector Front Thread (M16x2 - 6g)
  - Top Connector Thread (M10x1.5 - 6g)
  - PATTERN
  - Mirror1 (Features x YZ Plane (SIDE))
  - Mirror3 (Features x XY Plane (TOP))
  - EDGES
  - Flange Rad
  - Edge Clean up (1 mm)
  - End of Part



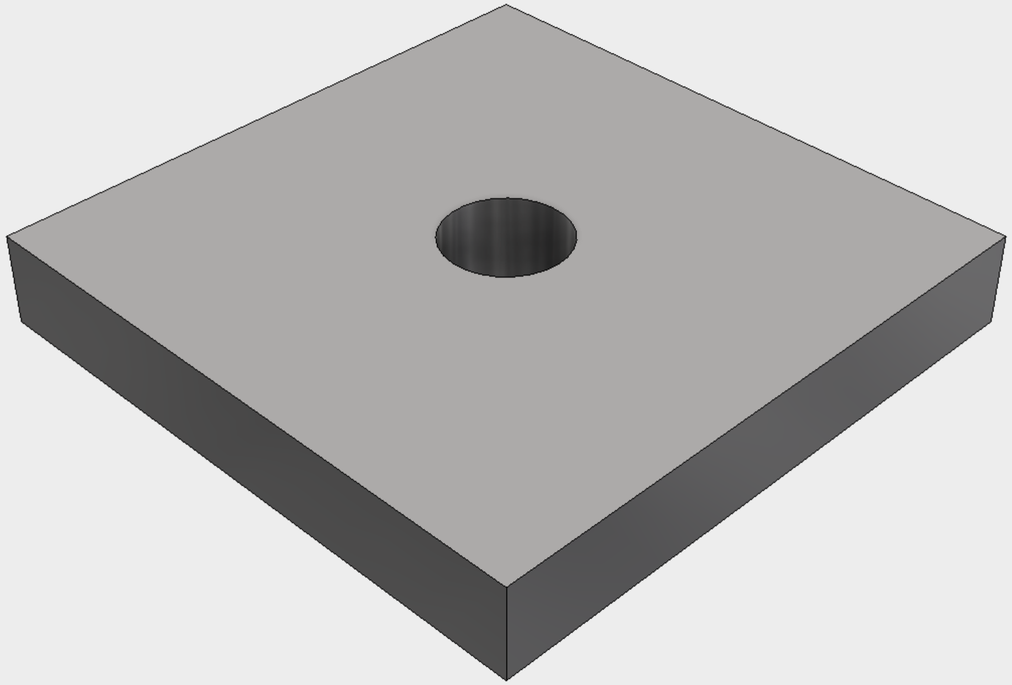
Multiple **Sketches**

Multiple **Features**

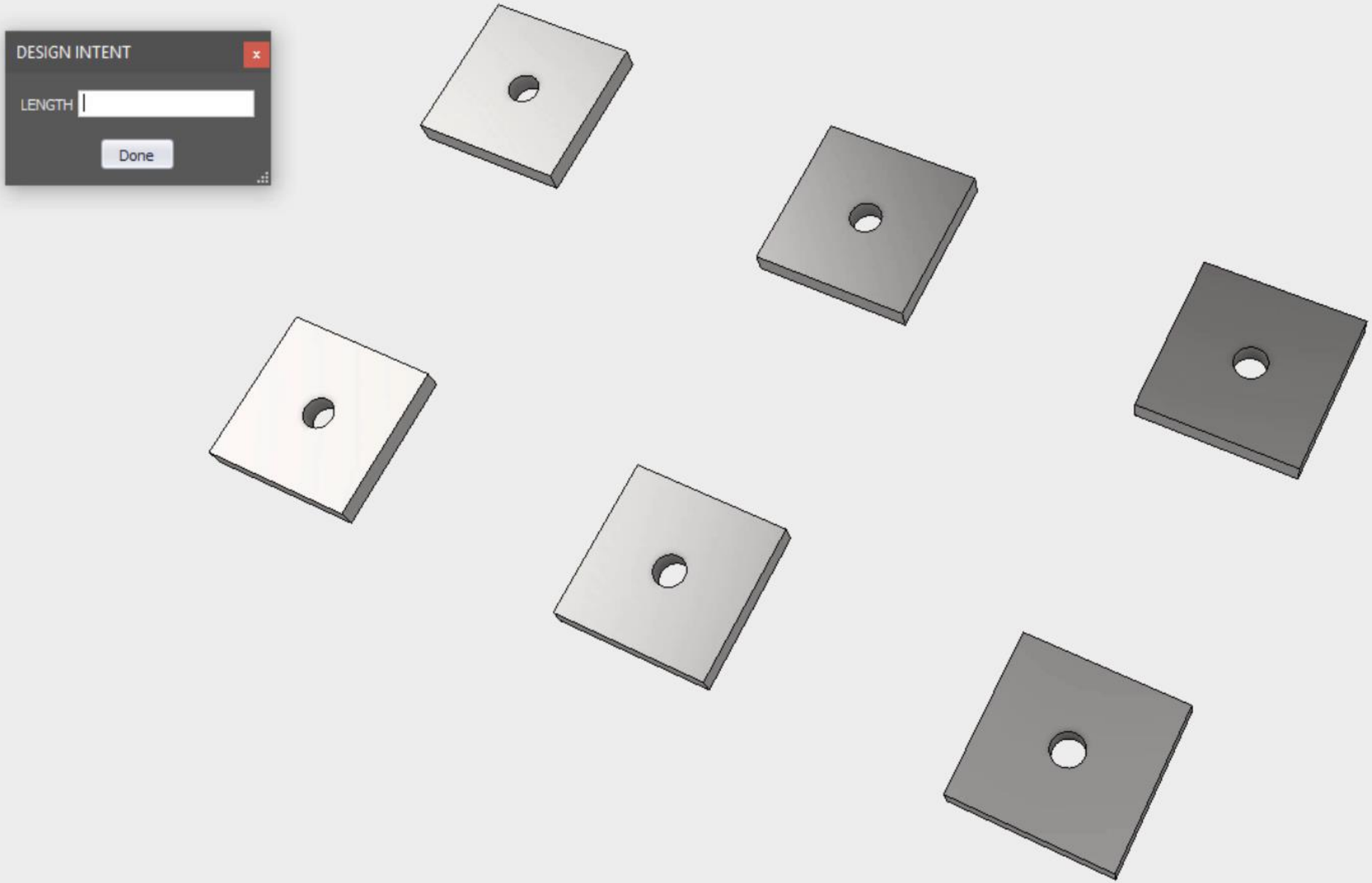
Multiple **Bodies**

Multiple **Updates**

1. No **unintended** Relationships
2. Relationships are kept to a **minimum**
3. All relationships are **planned** and **purposeful**
4. All relationships are **obvious** & **easily understood**



Width = n


DESIGN INTENT
LENGTH
Done

SET UP

Model template checklist

Use this checklist to make sure that you have a robust template for parts, assemblies and presentations.

General

- ☐ Create a Parameter naming schema
- ☐ Create a Feature naming schema

Application options

- ☐ Turn ‘Show Extended Names’ on.

Part template (And Sheet Metal Template)

- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes

Optional

- ☐ Edit Body and Surface prefixes
- ☐ Create a UCS base feature
- ☐ Create named parameters
- ☐ Create a Layout sketch

Assembly template (And Weldment Template)

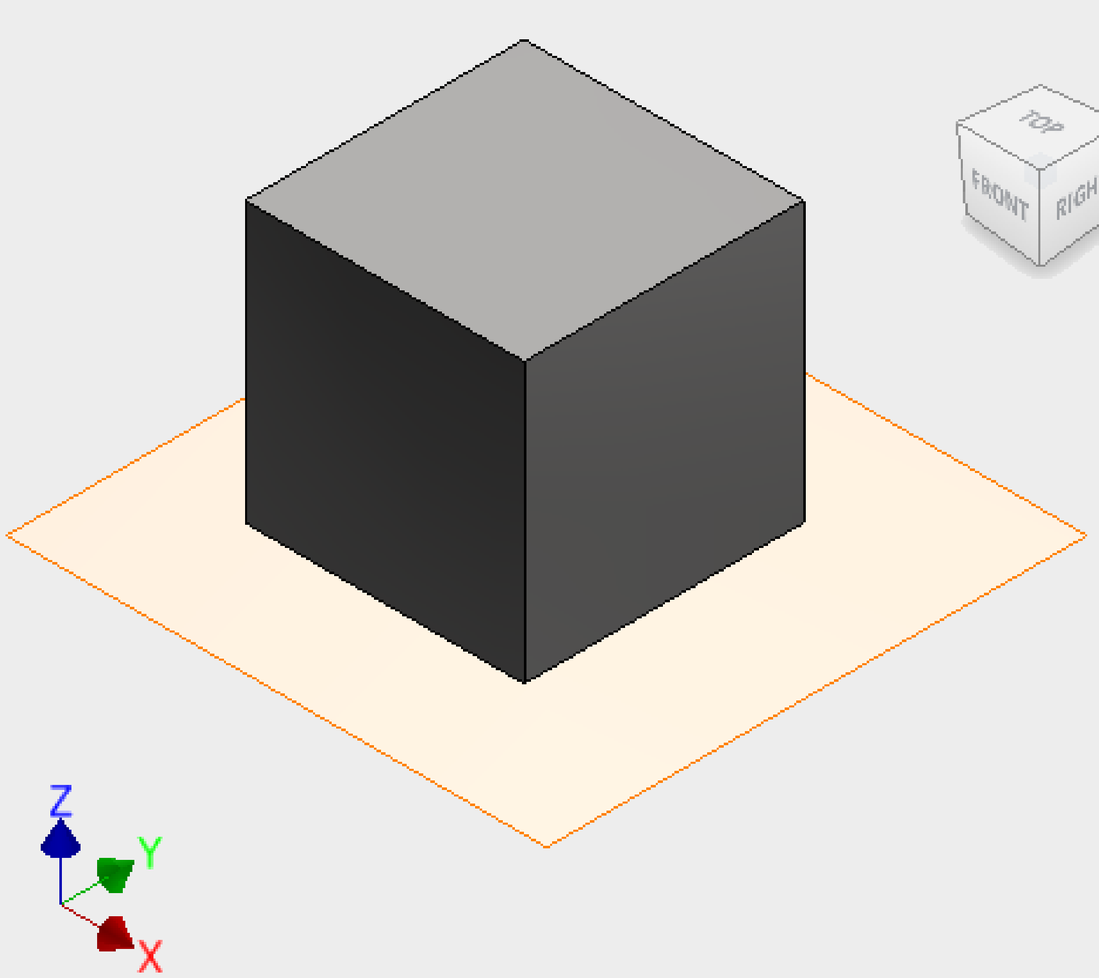
- ☐ Set the Viewcube orientation
- ☐ Set the default view
- ☐ Re-Name origin Planes

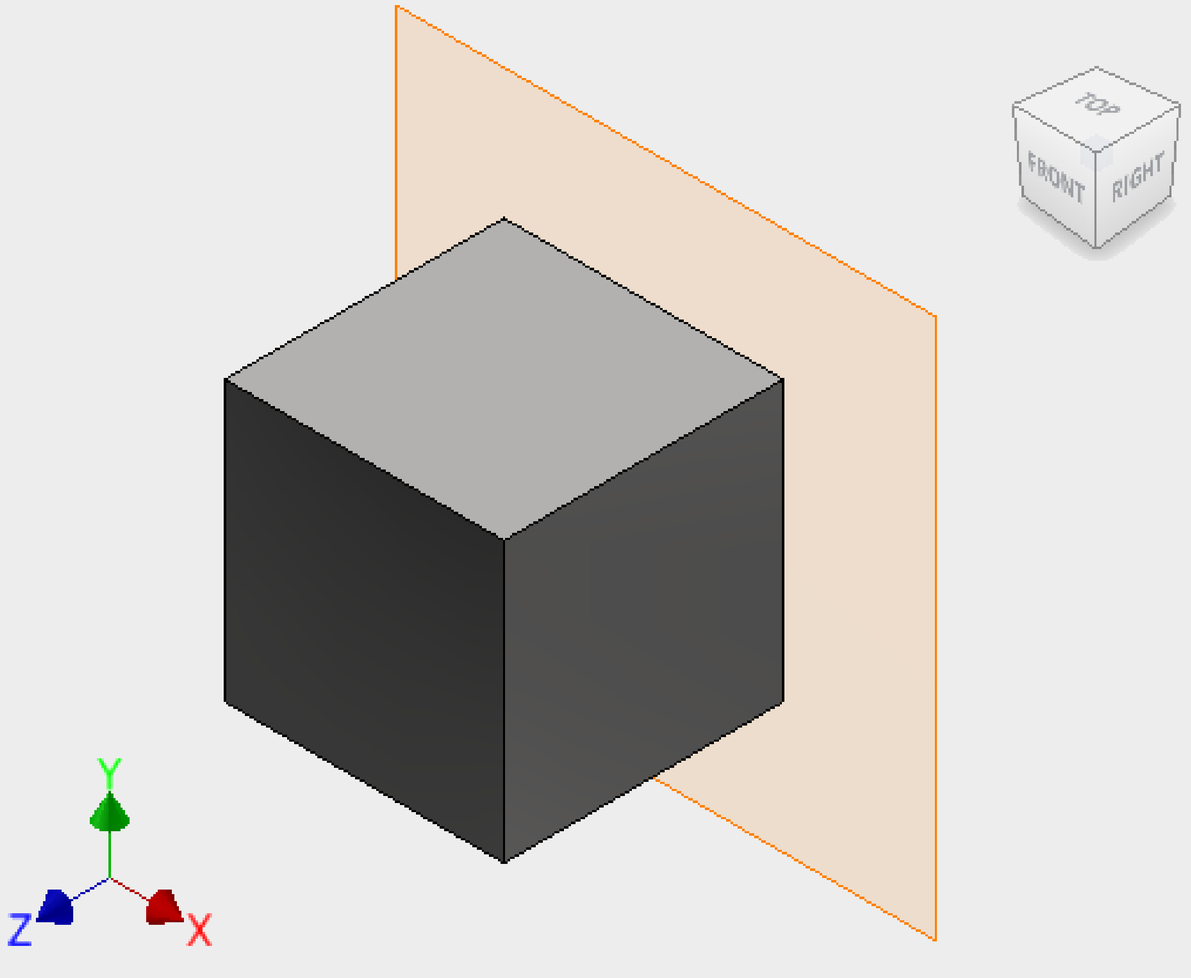
Optional

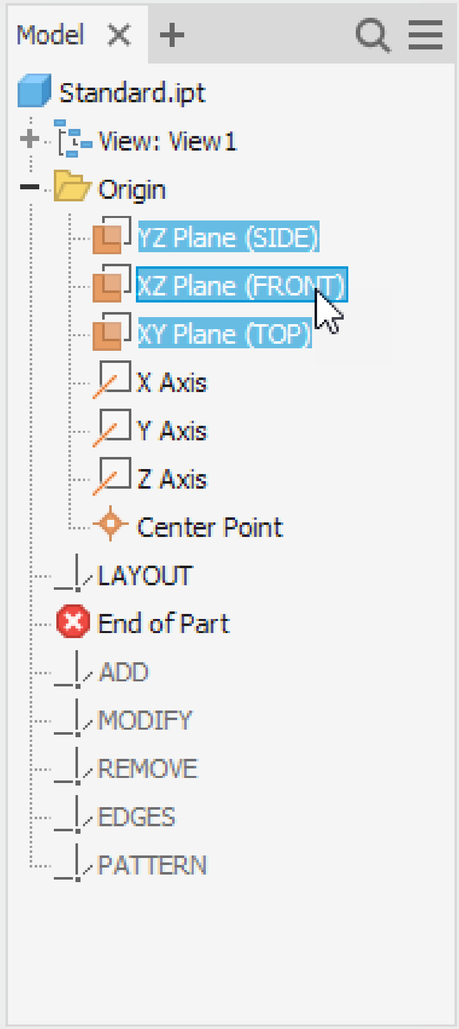
- ☐ Edit Body and Surface prefixes
- ☐ Create a UCS base feature
- ☐ Create named parameters

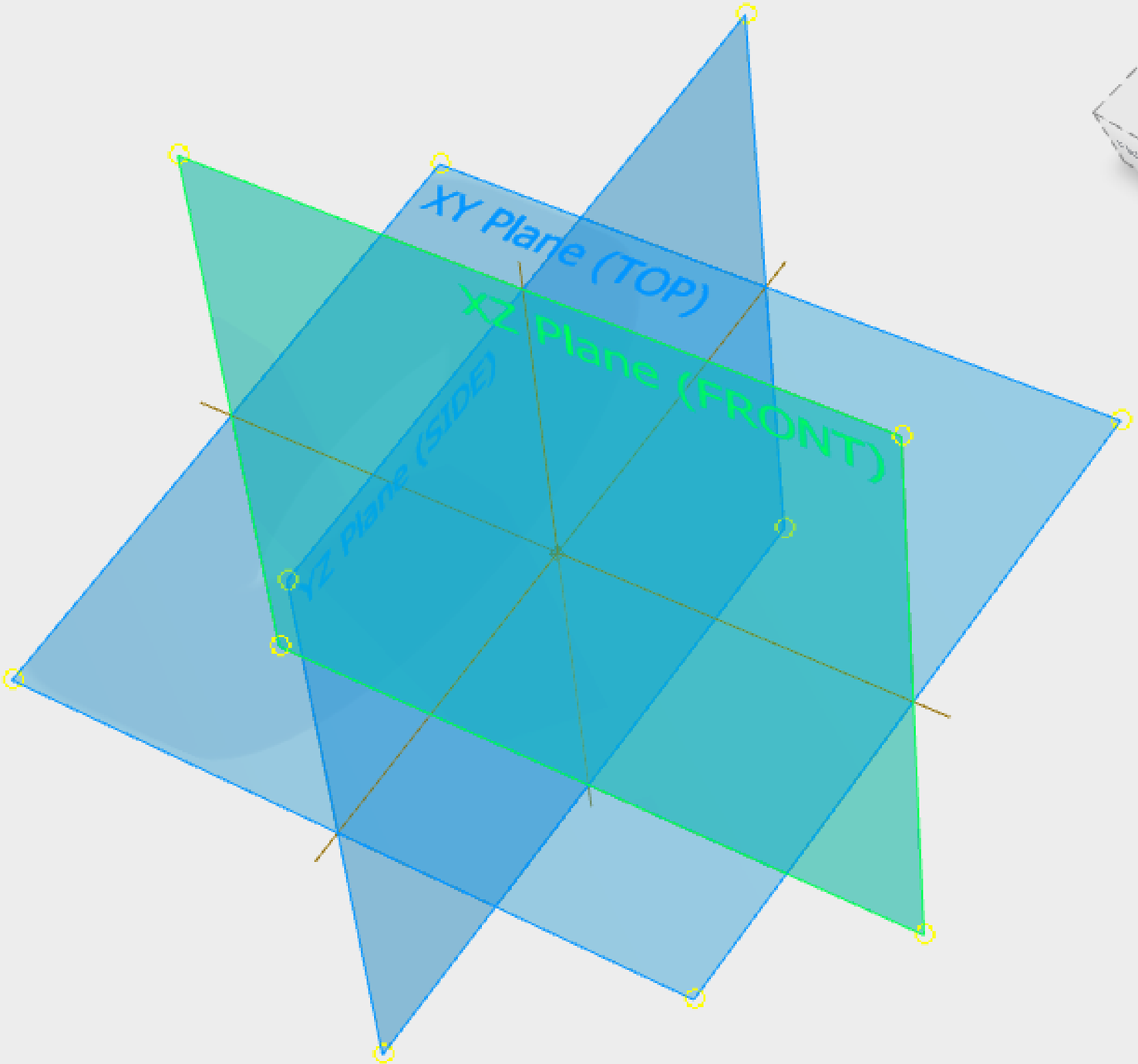
Presentation Template

- ☐ Set the Viewcube orientation
- ☐ Set the default view


TOP
FRONT
RIGHT
Z
Y
X


TOP
FRONT
RIGHT
Y
Z
X


Model
Standard.ipt
View: View1
Origin
YZ Plane (SIDE)
XZ Plane (FRONT)
XY Plane (TOP)
X Axis
Y Axis
Z Axis
Center Point
LAYOUT
End of Part
ADD
MODIFY
REMOVE
EDGES
PATTERN


XY Plane (TOP)
XZ Plane (FRONT)
YZ Plane (SIDE)


TOP
RIGHT

1. Case Sensitive
2. Start with a letter
3. Can Include Numbers
4. Cannot Contain spaces
5. Can contain ‘_’ and ‘:’

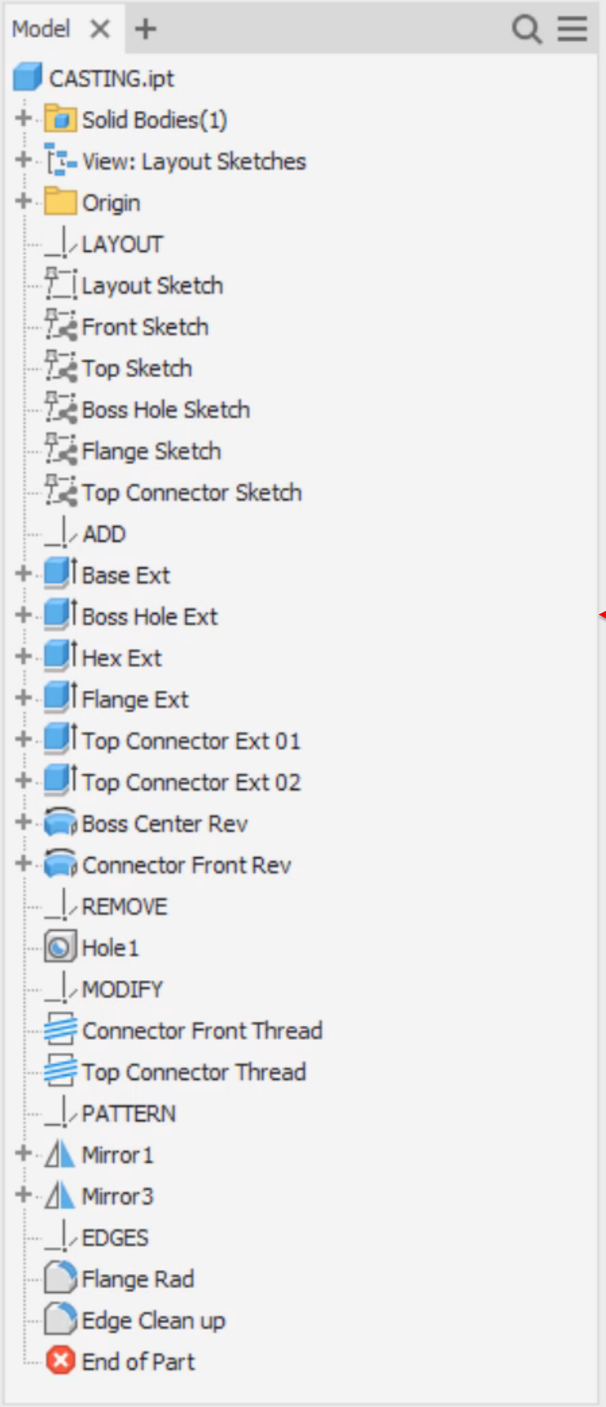
Examples

OverallWidth

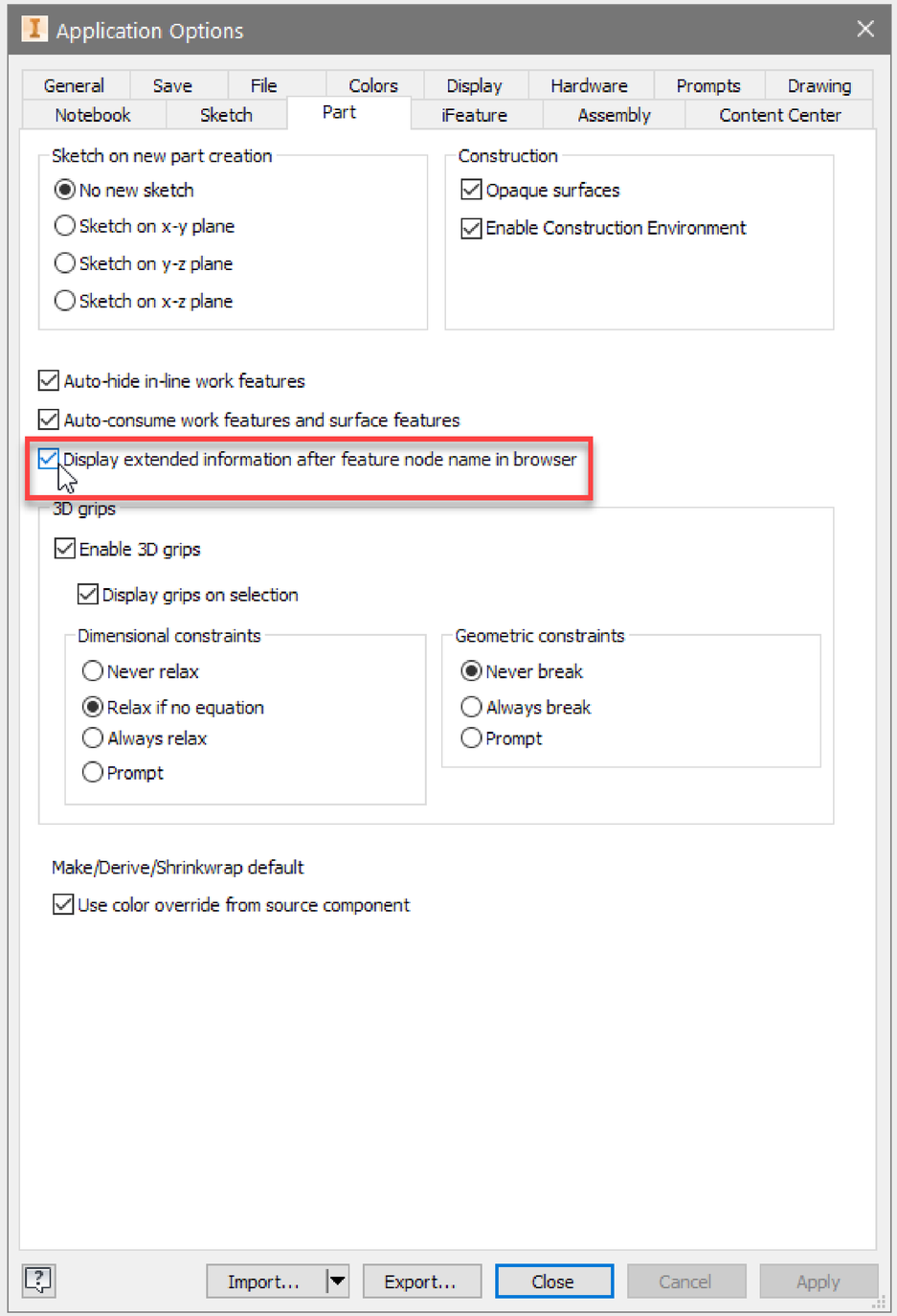
Overall_Width

OAwidth

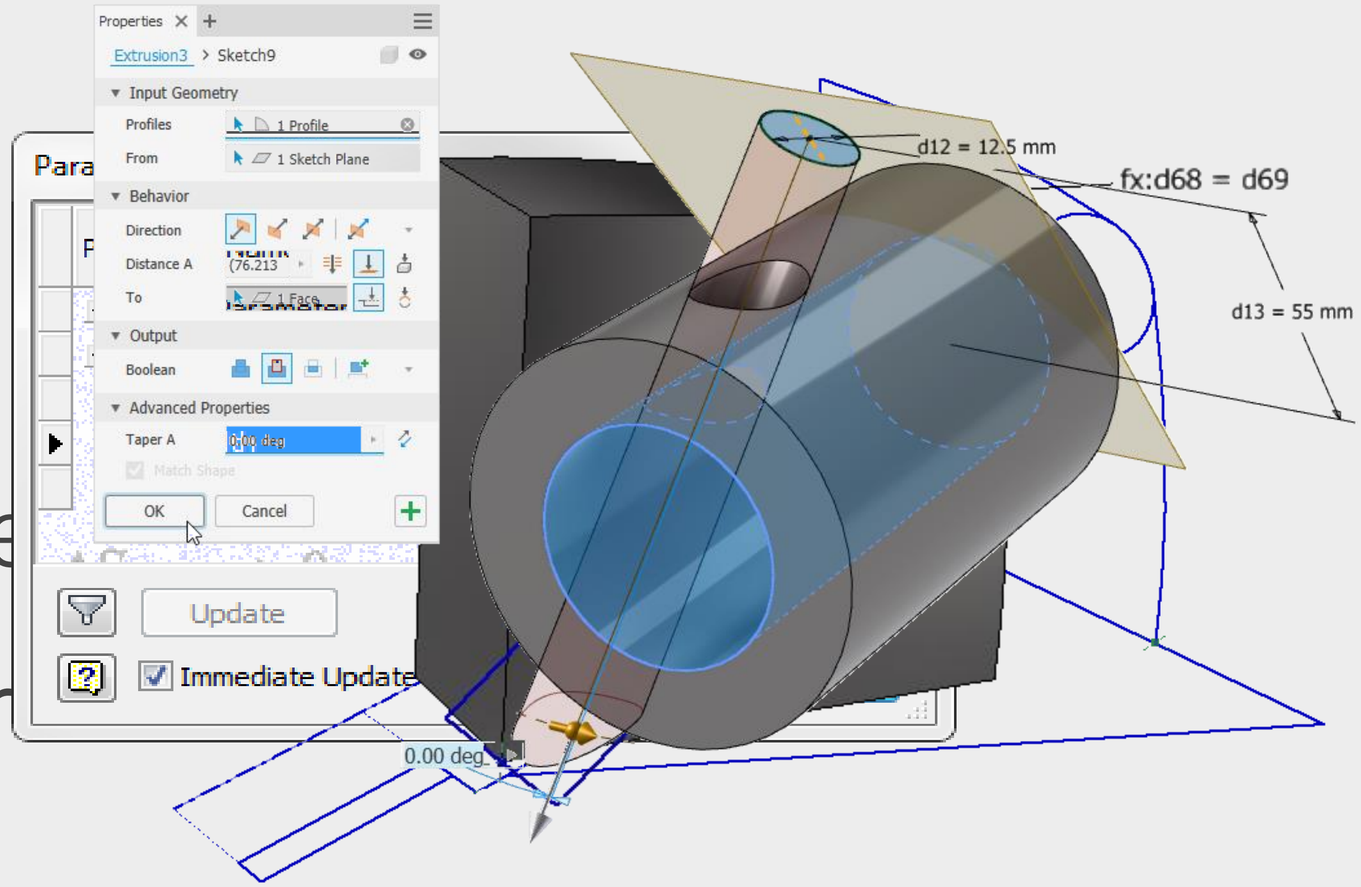
OA:Width



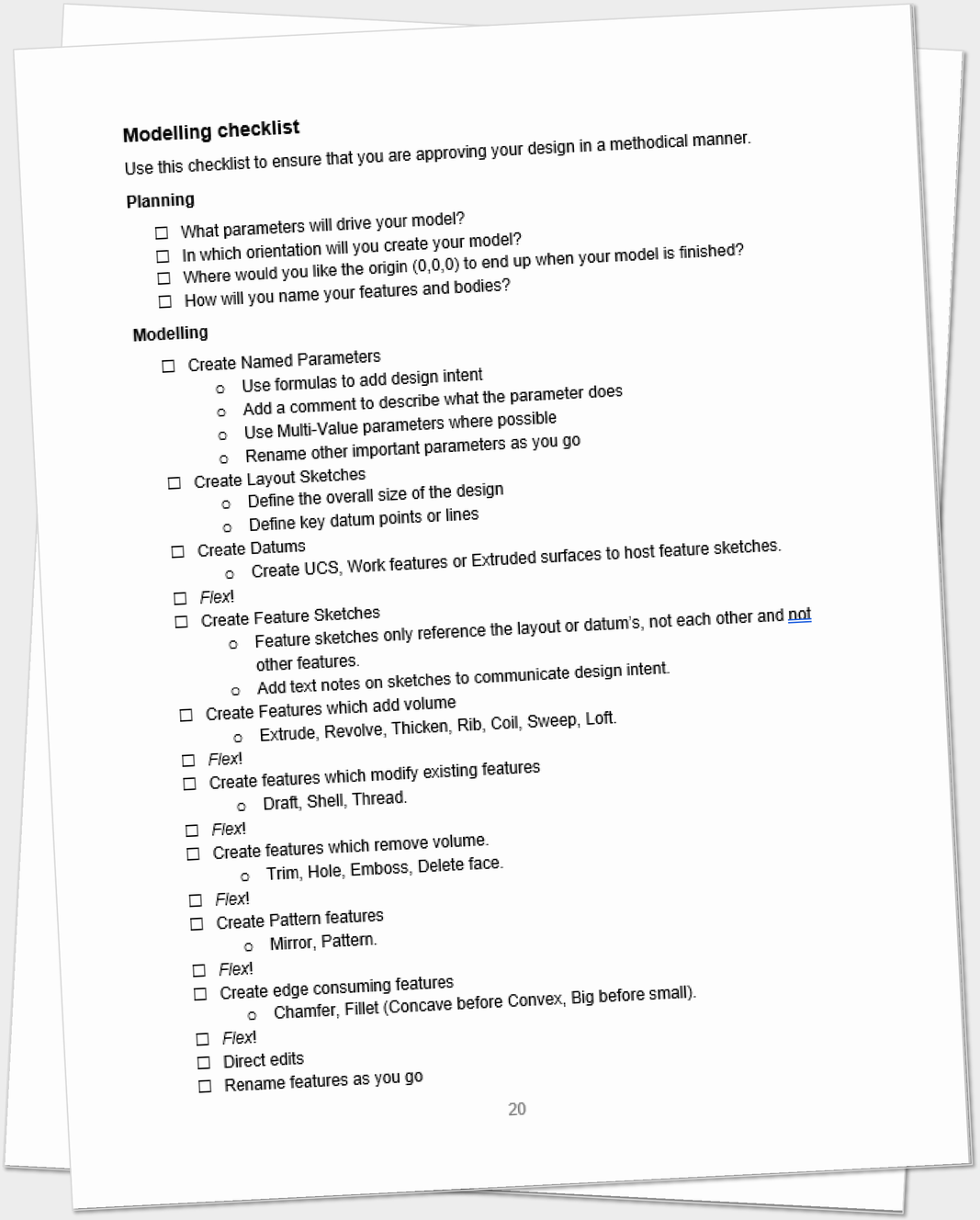
Additional information is added automatically.



1. Parametric
2. Sketch to Sketch
3. Sketch to Feature
4. Feature to Featur



WORK



Modelling checklist

Use this checklist to ensure that you are approving your design in a methodical manner.

Planning

- ☐ What parameters will drive your model?
- ☐ In which orientation will you create your model?
- ☐ Where would you like the origin (0,0,0) to end up when your model is finished?
- ☐ How will you name your features and bodies?

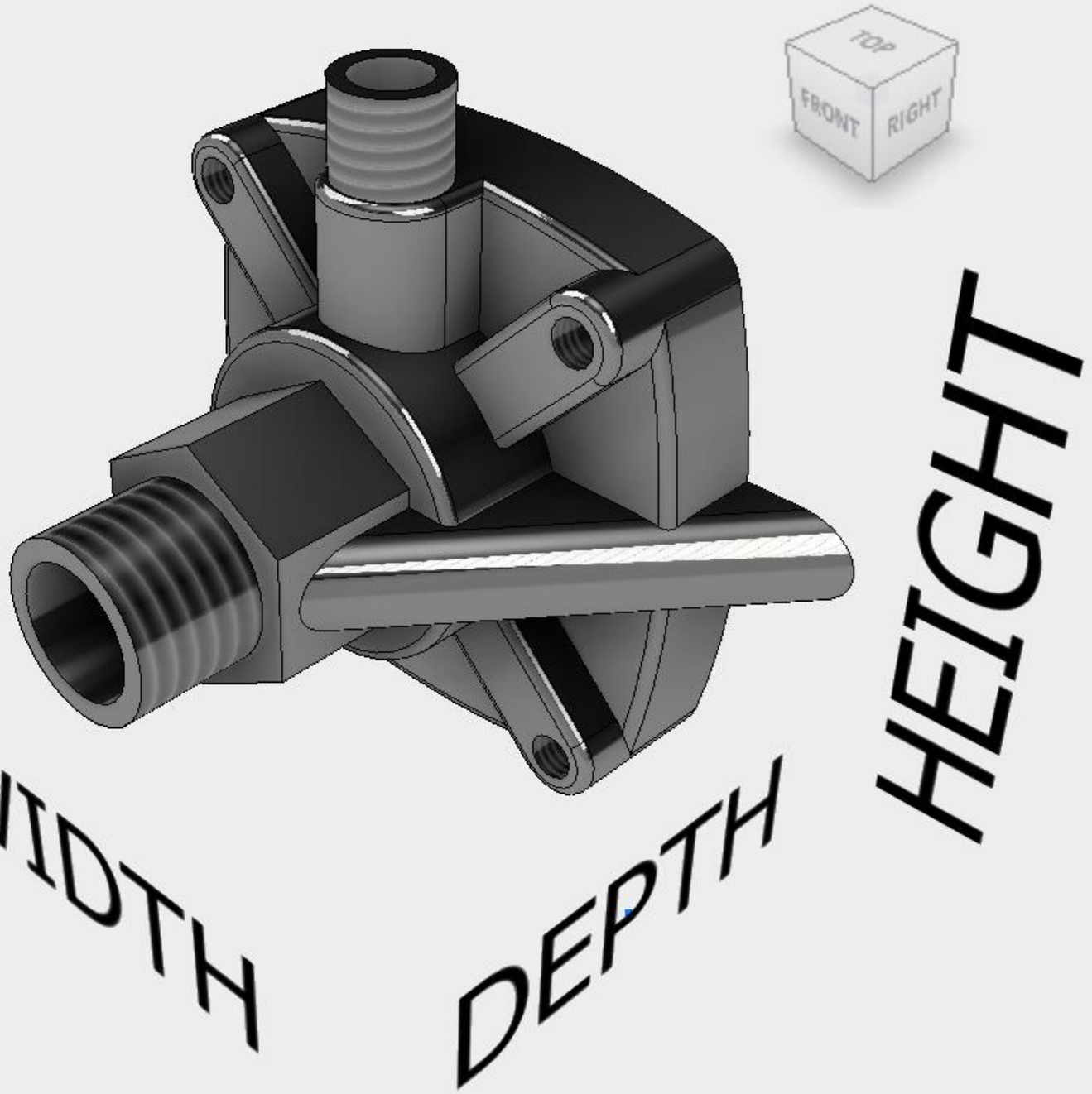
Modelling

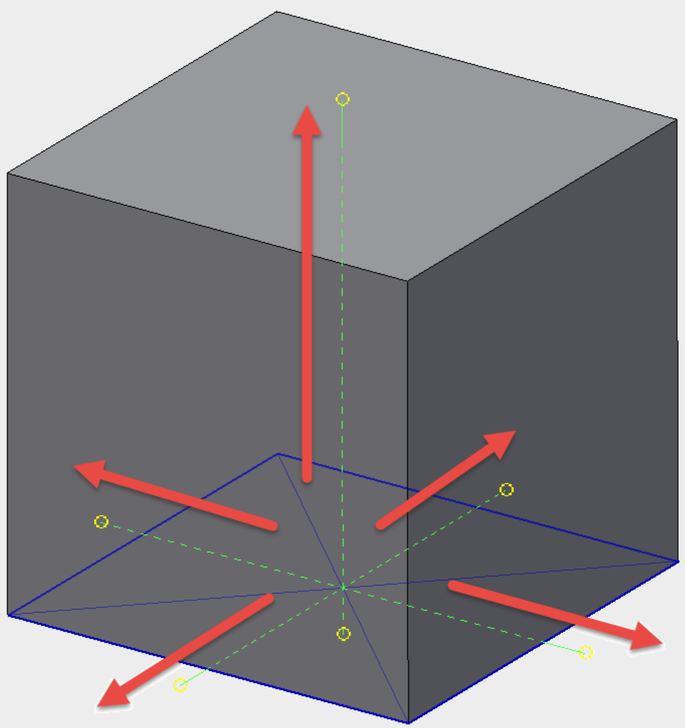
- ☐ Create Named Parameters
  - Use formulas to add design intent
  - Add a comment to describe what the parameter does
  - Use Multi-Value parameters where possible
  - Rename other important parameters as you go
- ☐ Create Layout Sketches
  - Define the overall size of the design
  - Define key datum points or lines
- ☐ Create Datums
  - Create UCS, Work features or Extruded surfaces to host feature sketches.
- ☐ *Flex*!
- ☐ Create Feature Sketches
  - Feature sketches only reference the layout or datum's, not each other and not other features.
  - Add text notes on sketches to communicate design intent.
- ☐ Create Features which add volume
  - Extrude, Revolve, Thicken, Rib, Coil, Sweep, Loft.
- ☐ *Flex*!
- ☐ Create features which modify existing features
  - Draft, Shell, Thread.
- ☐ *Flex*!
- ☐ Create features which remove volume.
  - Trim, Hole, Emboss, Delete face.
- ☐ *Flex*!
- ☐ Create Pattern features
  - Mirror, Pattern.
- ☐ *Flex*!
- ☐ Create edge consuming features
  - Chamfer, Fillet (Concave before Convex, Big before small).
- ☐ *Flex*!
- ☐ Direct edits
- ☐ Rename features as you go

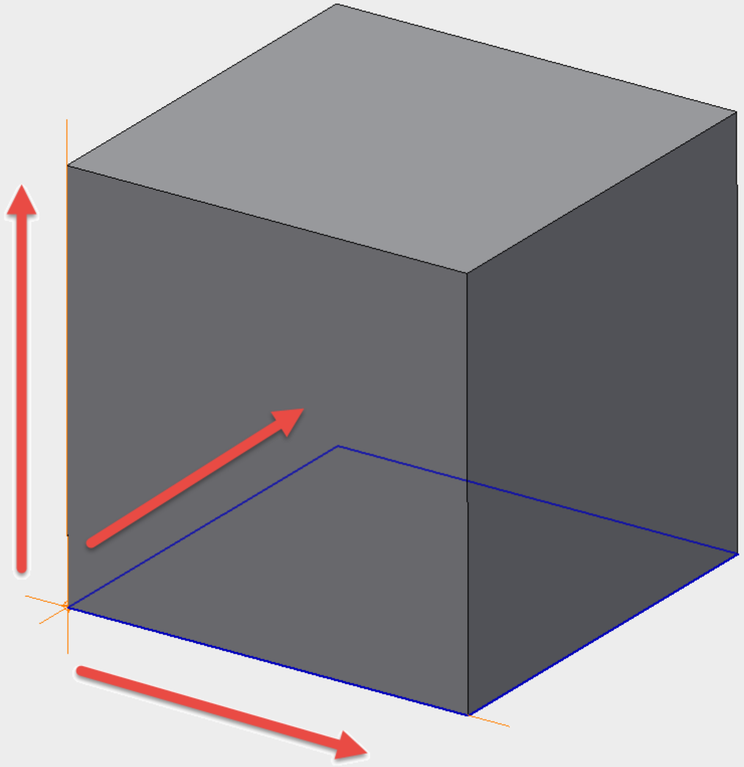
Before you start – **STOP**! Planning is key.

Observe

- Orientation
- Origin
- Feature names
- Parameter names
- Change


TOP
FRONT
RIGHT
WIDTH
DEPTH
HEIGHT





Parameters

Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Expo	Comment
Model Parameters									
User Parameters									
OAheight		mm	0 mm	0.000000		0.000000	☑	☐	
OAwidth		mm	0 mm	0.000000		0.000000	☑	☐	
OAdepth		mm	0 mm	0.000000		0.000000	☑	☐	

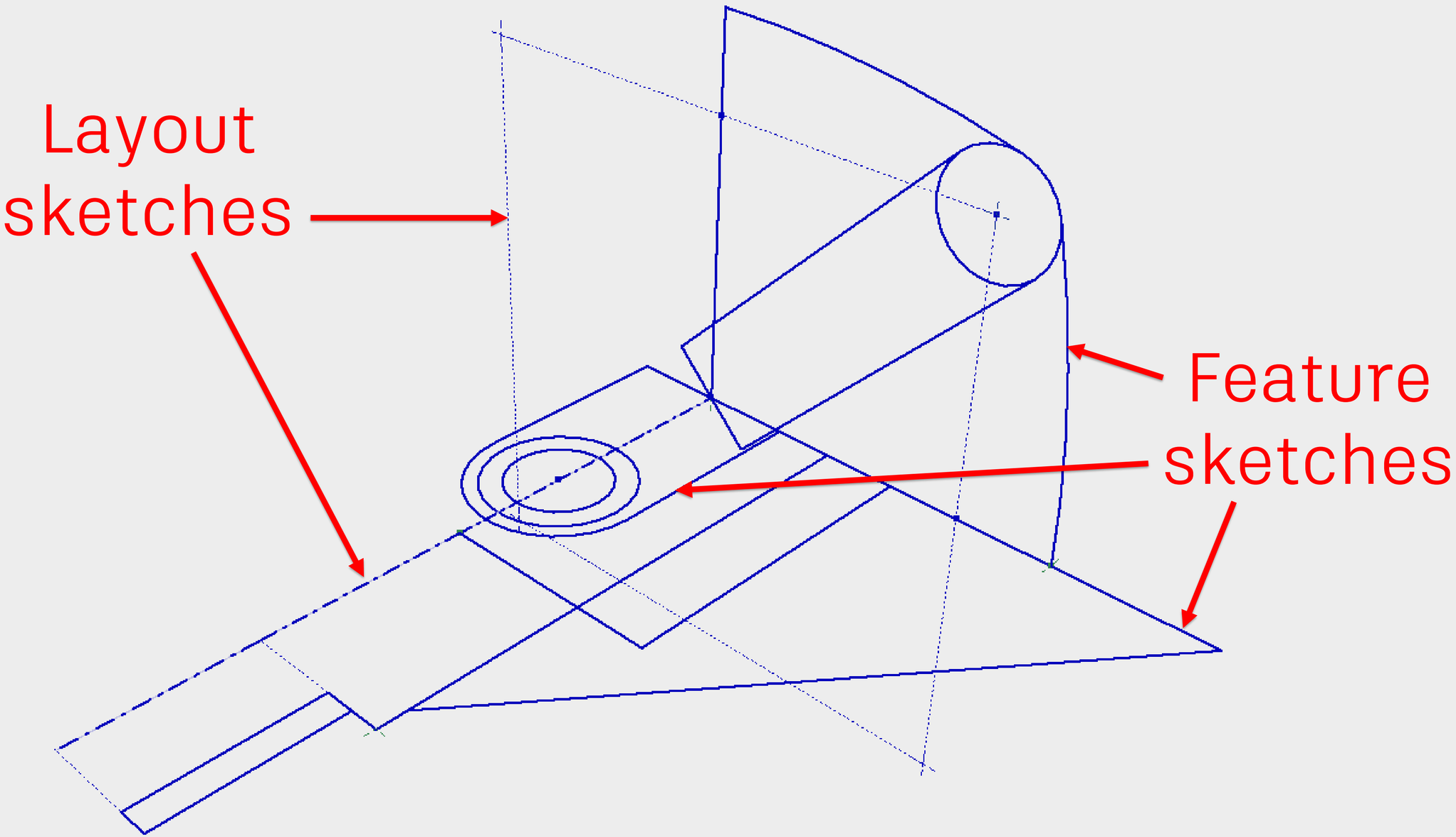
Add Numeric | Update | Purge Unused

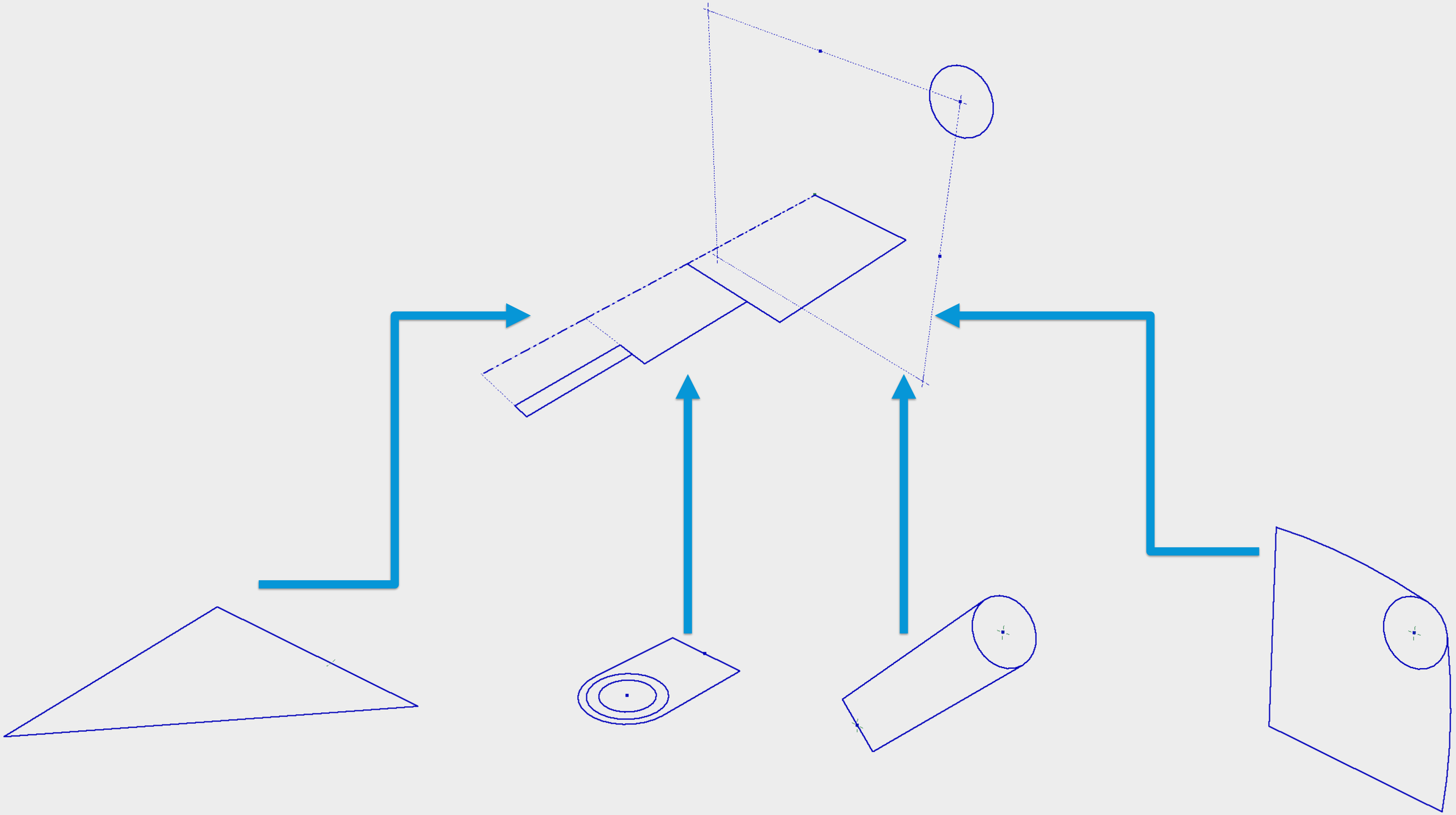
Link | Immediate Update

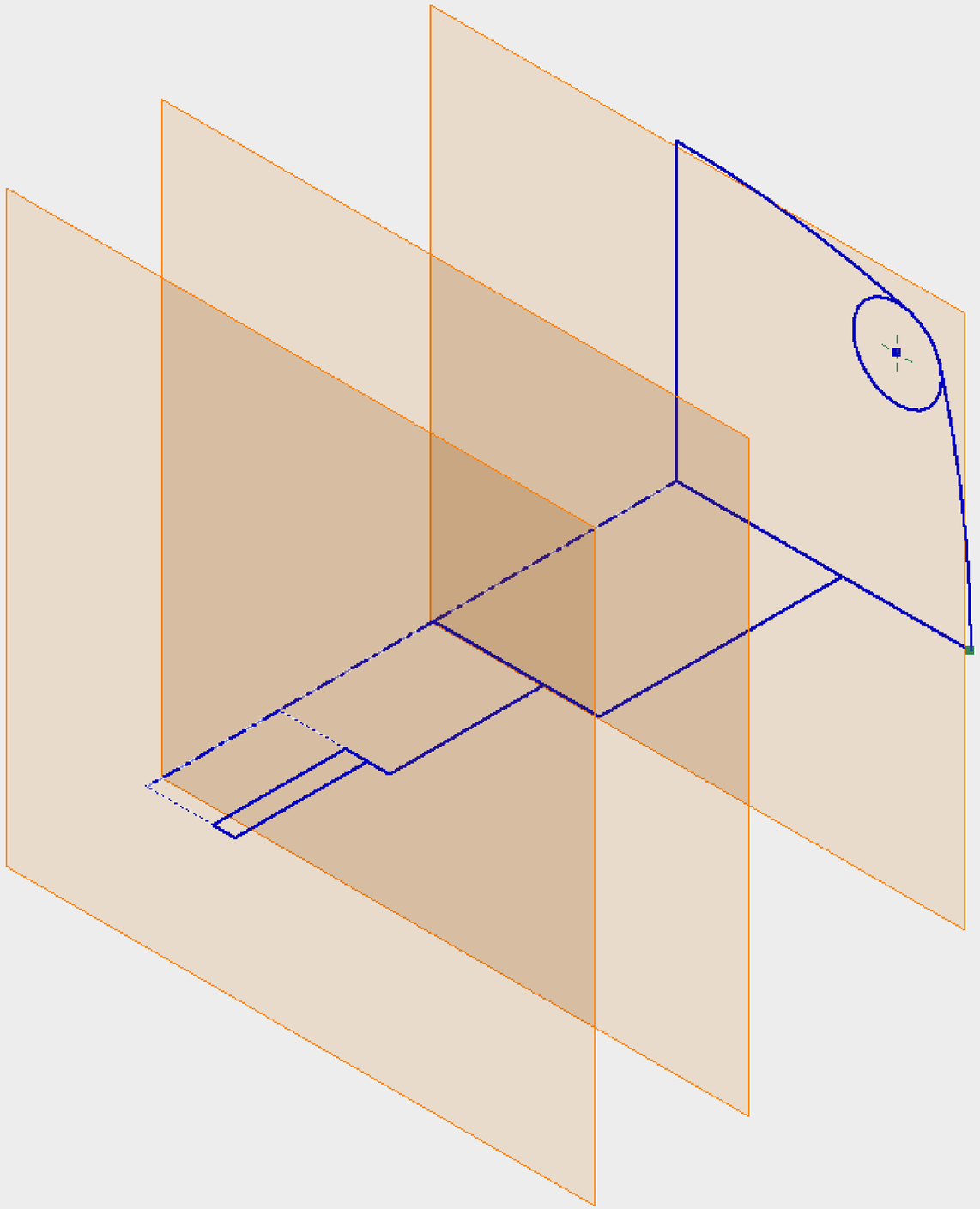
Reset Tolerance

<< Less

Done


Layout
sketches
Feature
sketches





Add >

Extrude

Revolve

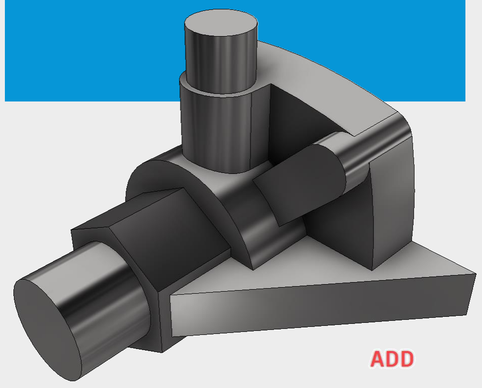
Thicken

Rib

Coil

Sweep

Loft



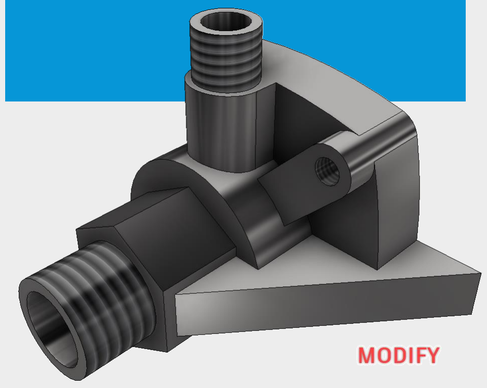
Modify >

Draft

Shell

Thread

(Fillet)



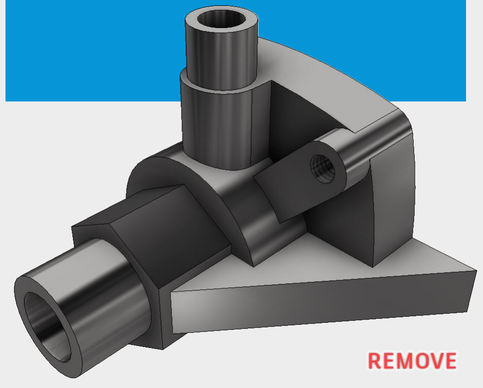
Remove >

Trim

Hole

Emboss

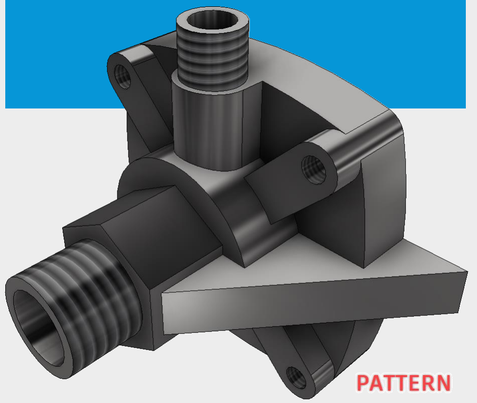
Delete face



Pattern >

Mirror

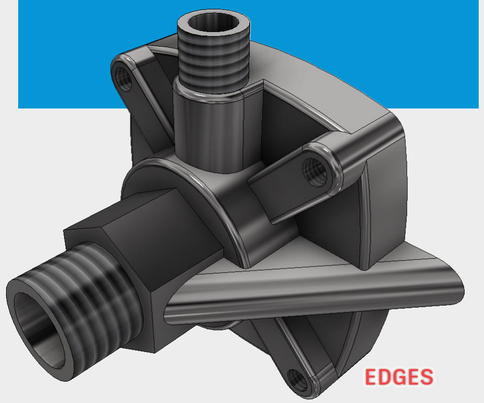
Pattern



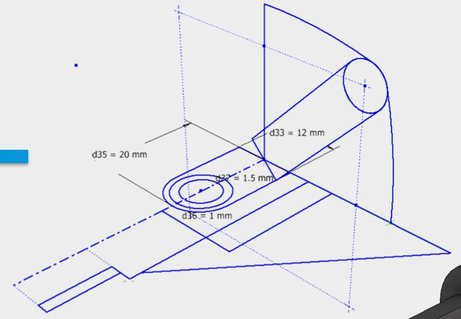
Edges

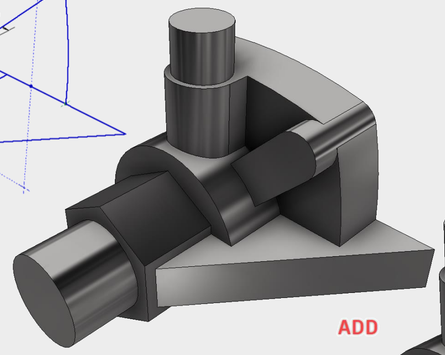
Chamfer

Fillet

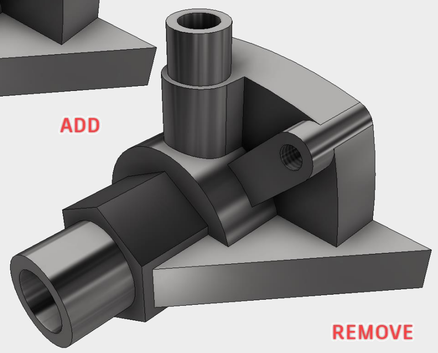


- CASTING.ipt
  - Solid Bodies(1)
  - View: Master
  - Origin
  - LAYOUT
    - Layout Sketch
    - Front Sketch
    - Top Sketch
    - Boss Hole Sketch
    - Flange Sketch
    - Top Connector Sketch
  - ADD
    - Base Ext (New Solid x BaseDepth)
    - Boss Hole Ext (Join x BaseDepth + BossHoleDepth)
    - Hex Ext (Join x d21)
    - Flange Ext (Join x FlangeThick)
    - Top Connector Ext 01 (Join To)
    - Top Connector Ext 02 (Join x 35 mm)
    - Boss Center Rev (Join Full)
    - Connector Front Rev (Join Full)
  - REMOVE
    - Hole1 (M5x0.8 x 3 mm ThreadDepth x 5 mm Depth)
  - MODIFY
    - Connector Front Thread (M16x2 - 6g)
    - Top Connector Thread (M10x1.5 - 6g)
  - PATTERN
    - Mirror1 (Features x YZ Plane (SIDE))
    - Mirror3 (Features x XY Plane (TOP))
  - EDGES
    - Flange Rad
    - Edge Clean up (1 mm)
    - End of Part

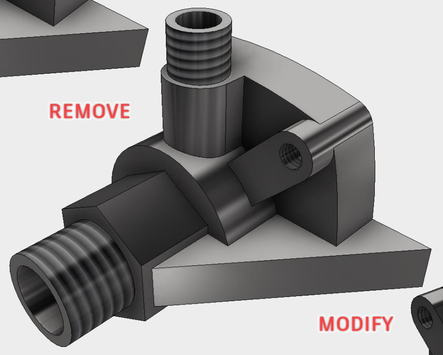




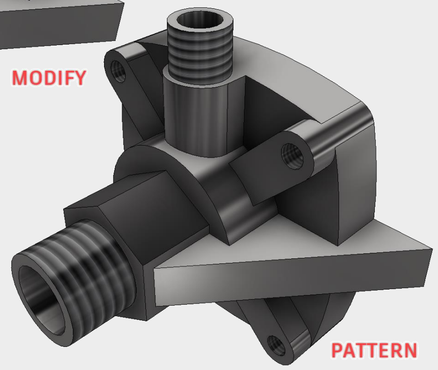
ADD



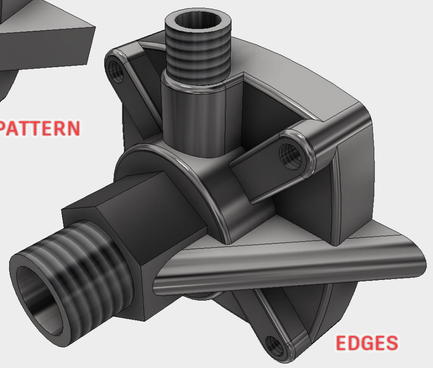
REMOVE



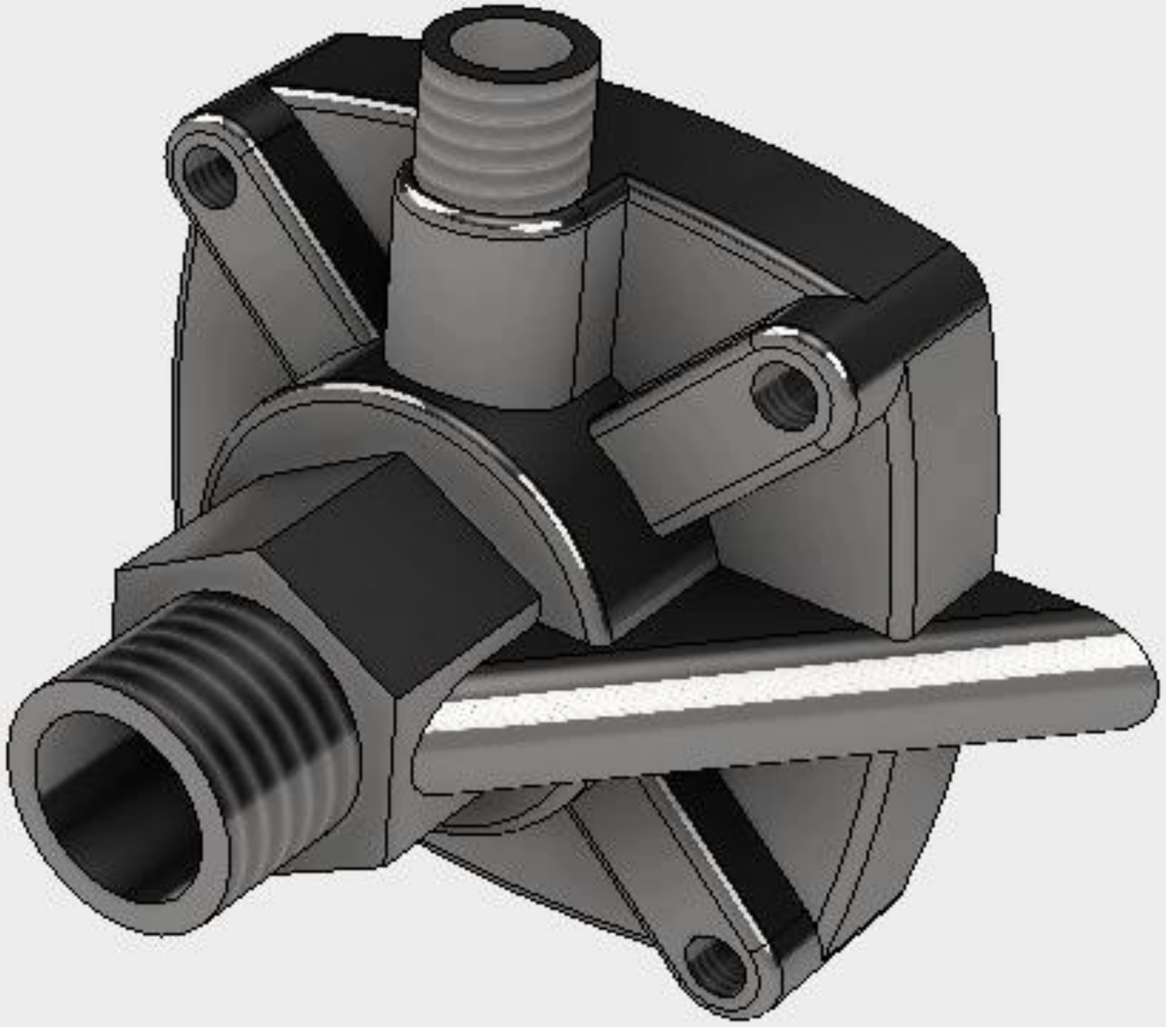
MODIFY

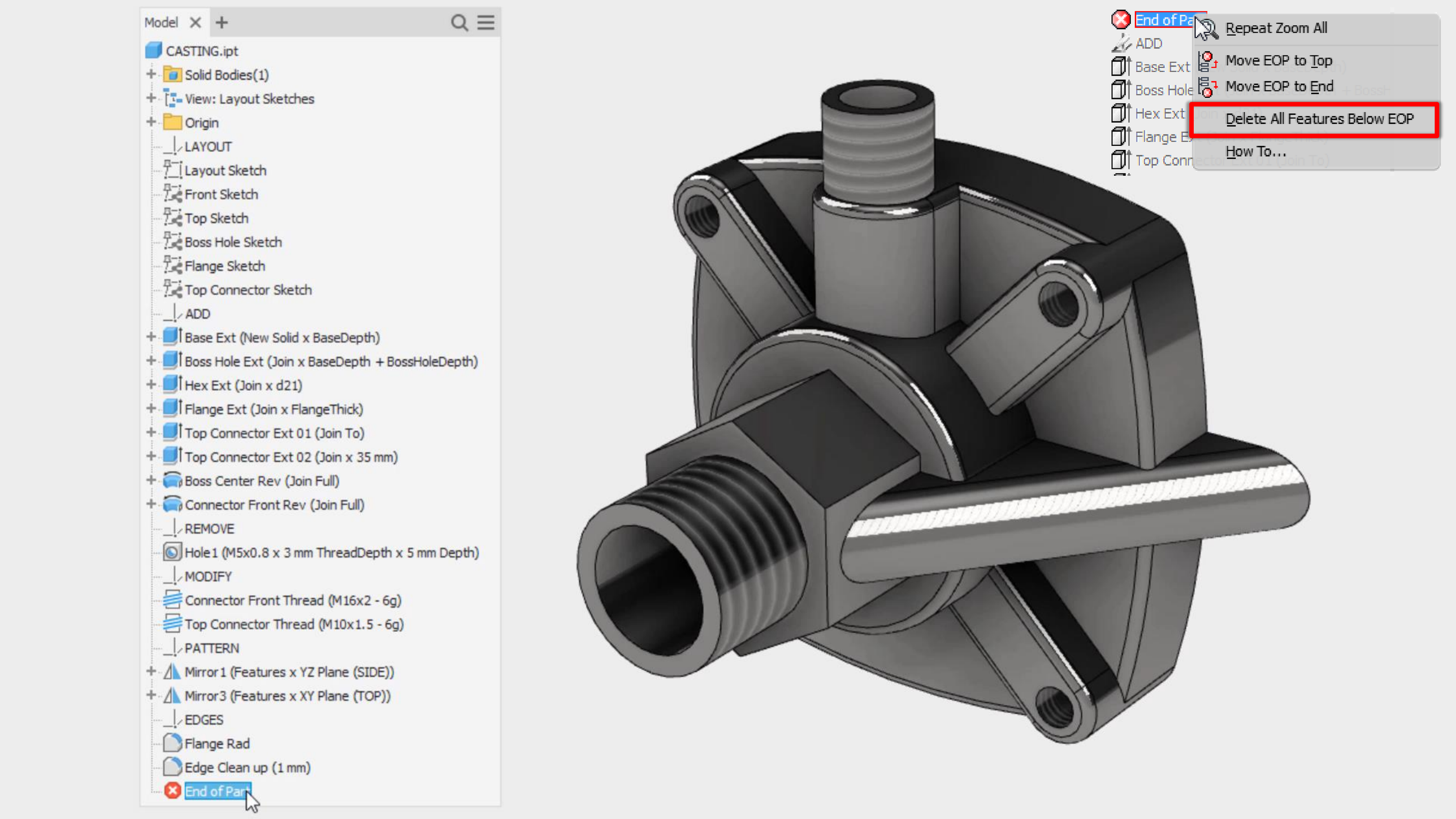


PATTERN

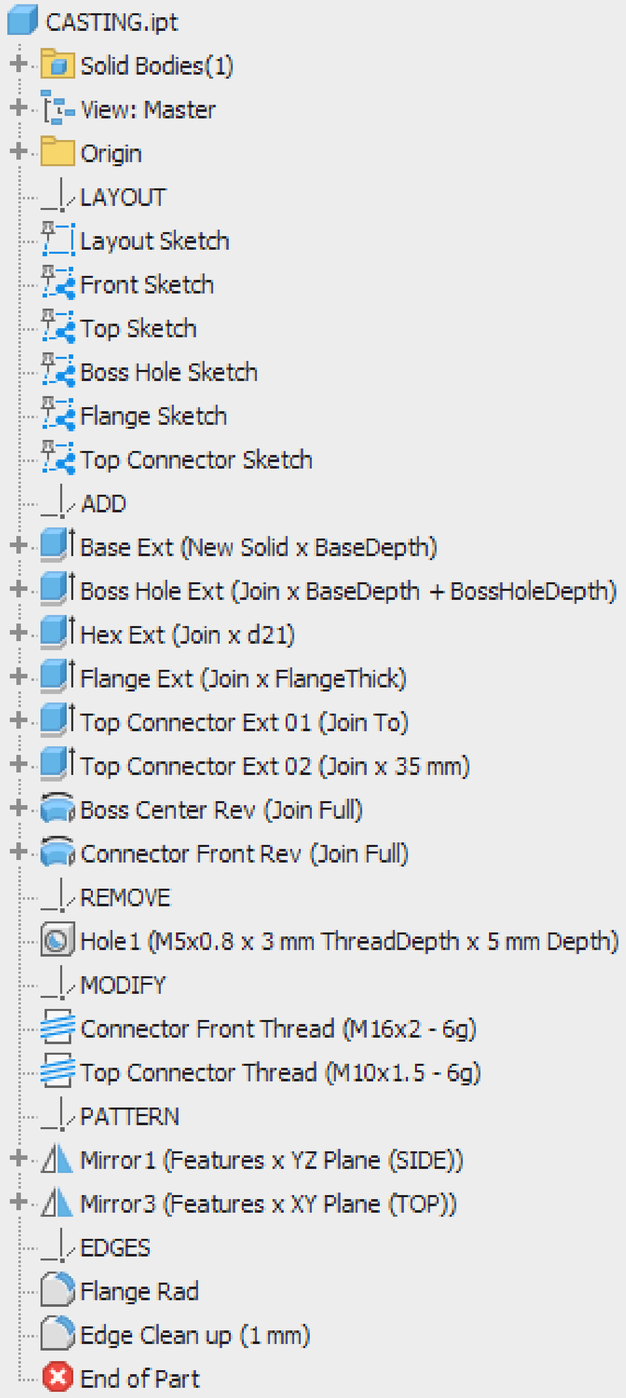


EDGES



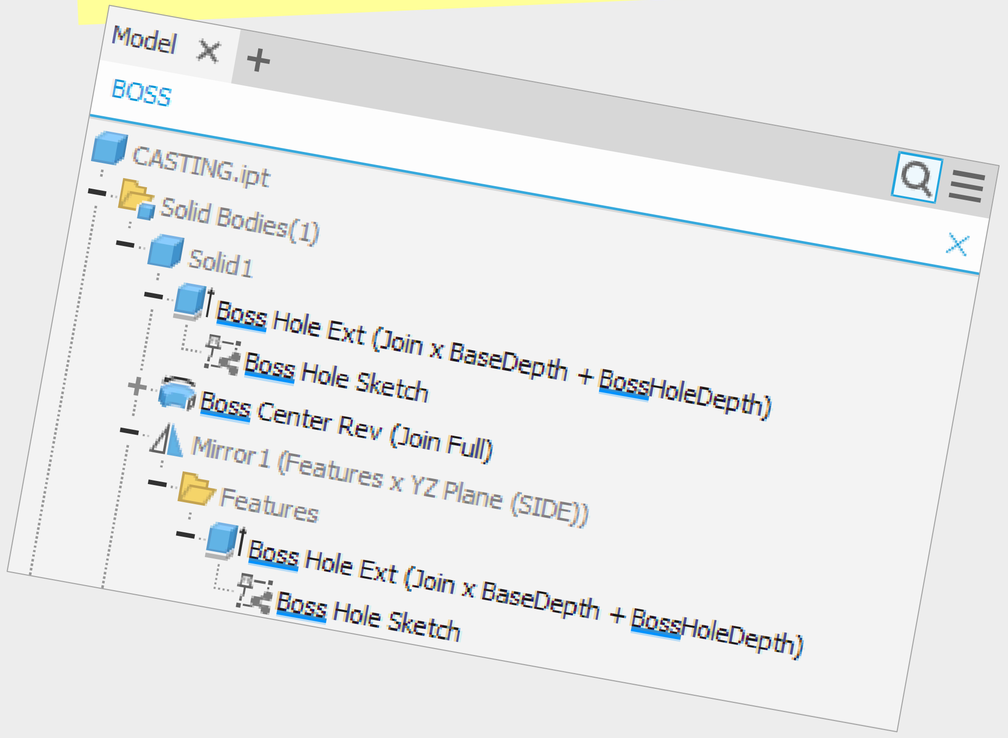

Model
CASTING.ipt
Solid Bodies(1)
View: Layout Sketches
Origin
LAYOUT
Layout Sketch
Front Sketch
Top Sketch
Boss Hole Sketch
Flange Sketch
Top Connector Sketch
ADD
Base Ext (New Solid x BaseDepth)
Boss Hole Ext (Join x BaseDepth + BossHoleDepth)
Hex Ext (Join x d21)
Flange Ext (Join x FlangeThick)
Top Connector Ext 01 (Join To)
Top Connector Ext 02 (Join x 35 mm)
Boss Center Rev (Join Full)
Connector Front Rev (Join Full)
REMOVE
Hole1 (M5x0.8 x 3 mm ThreadDepth x 5 mm Depth)
MODIFY
Connector Front Thread (M16x2 - 6g)
Top Connector Thread (M10x1.5 - 6g)
PATTERN
Mirror1 (Features x YZ Plane (SIDE))
Mirror3 (Features x XY Plane (TOP))
EDGES
Flange Rad
Edge Clean up (1 mm)
End of Part
End of Pa
ADD
Base Ext
Boss Hole
Hex Ext
Flange E
Top Conn
Repeat Zoom All
Move EOP to Top
Move EOP to End
Delete All Features Below EOP
How To...

DOCUMENT



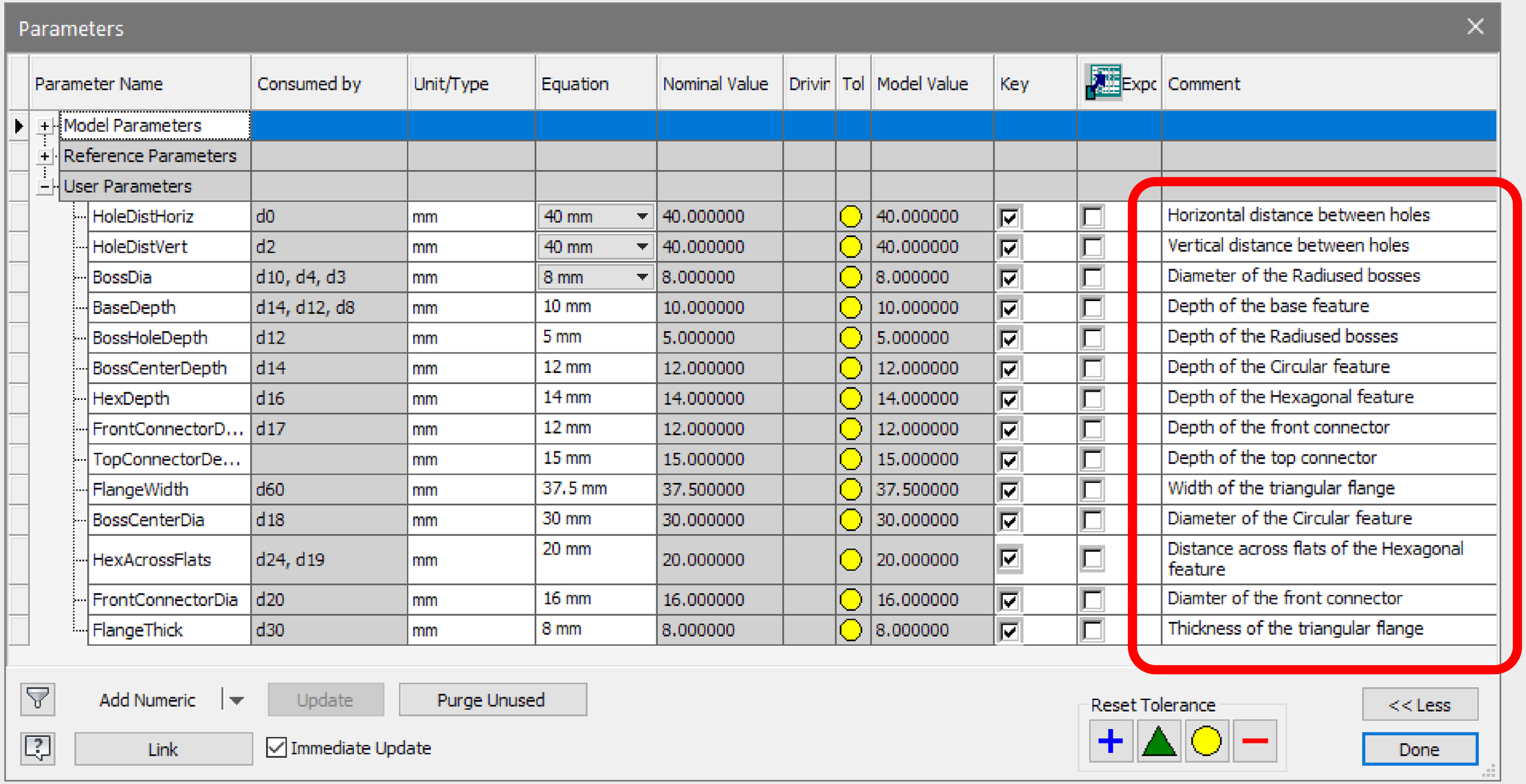
Example feature naming

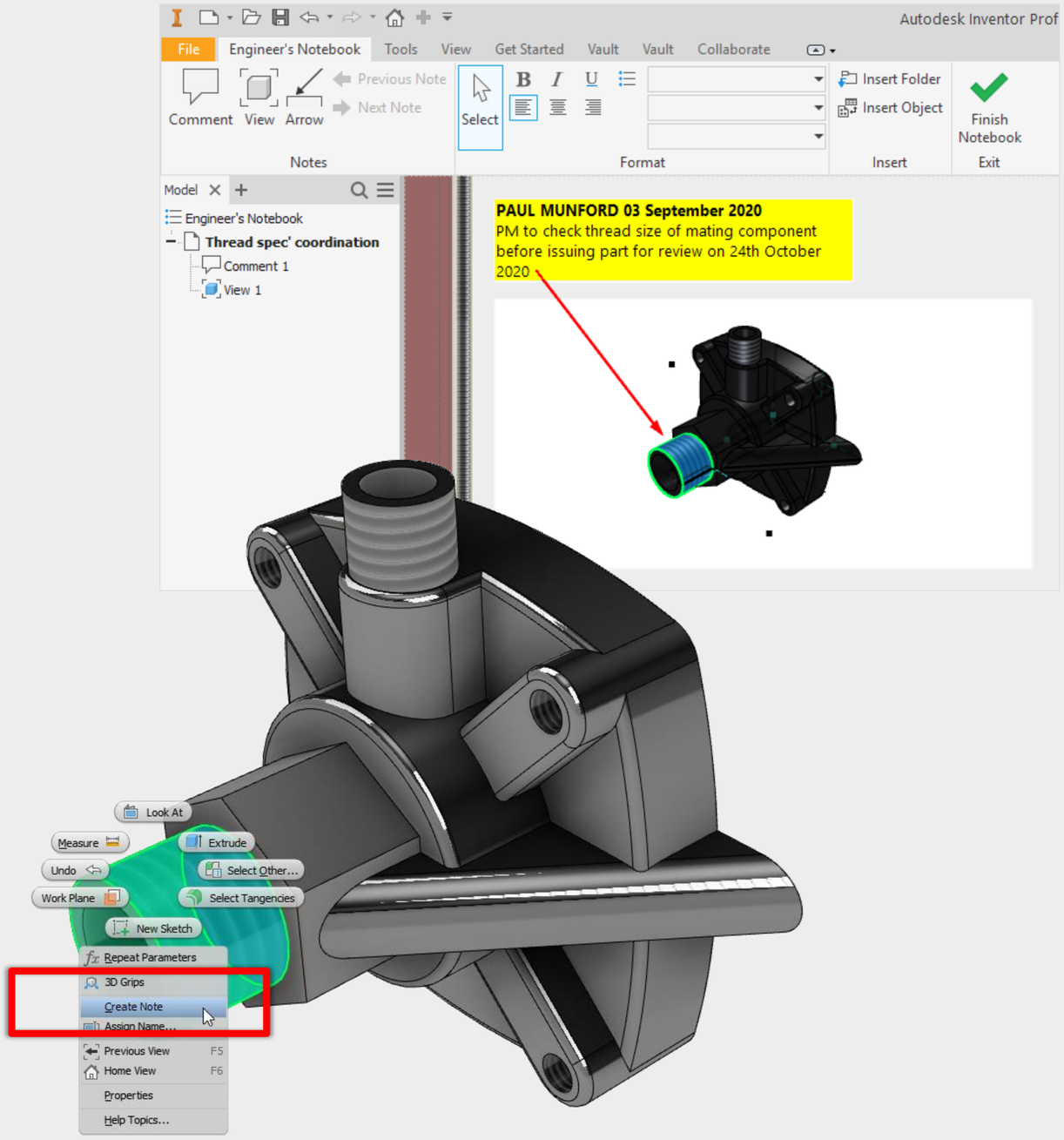
- **Boss** (Solid)
  - **Boss Hole** (A Hole in The Boss)
    - **Boss Ext** (The Extrusion that creates the Boss)
      - **Boss Sketch** (The Sketch for the Boss features)

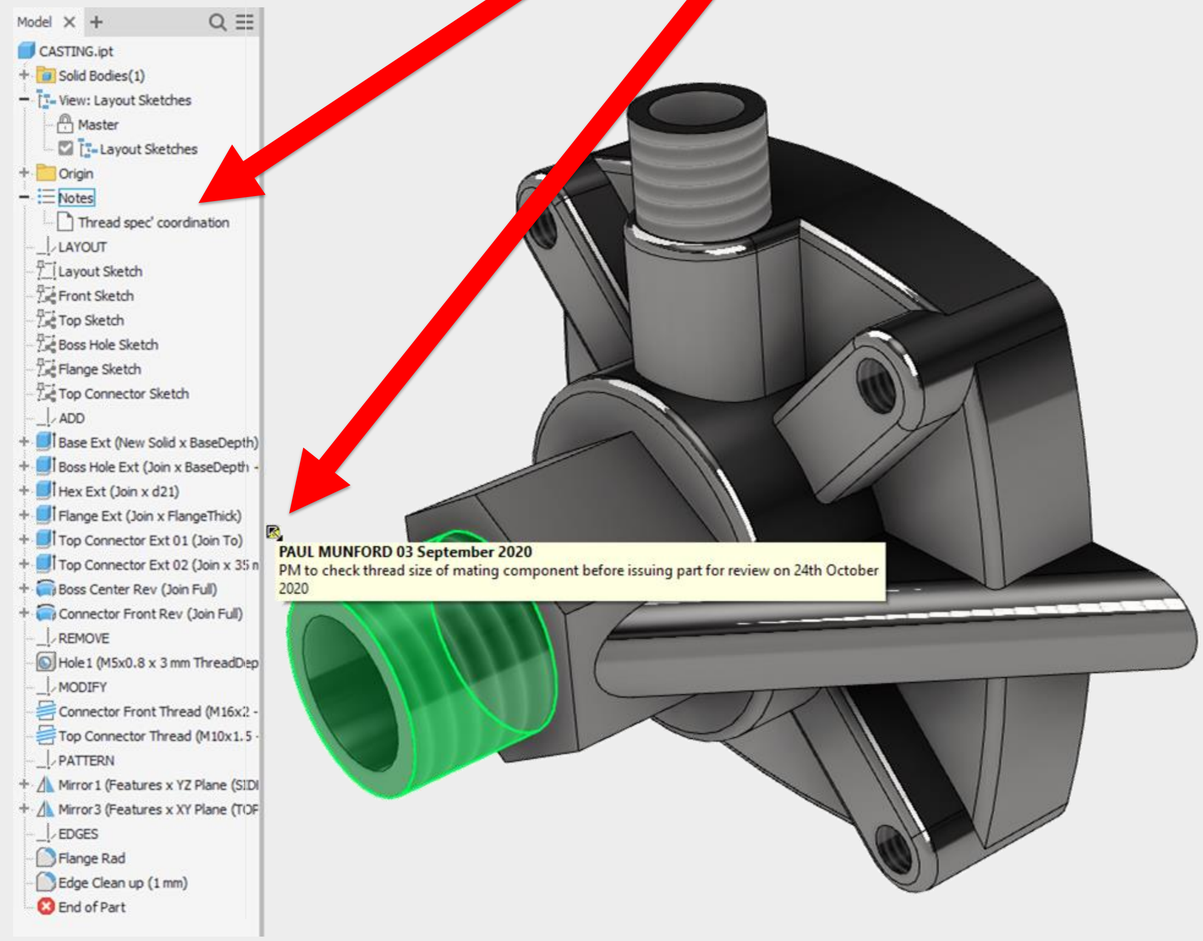


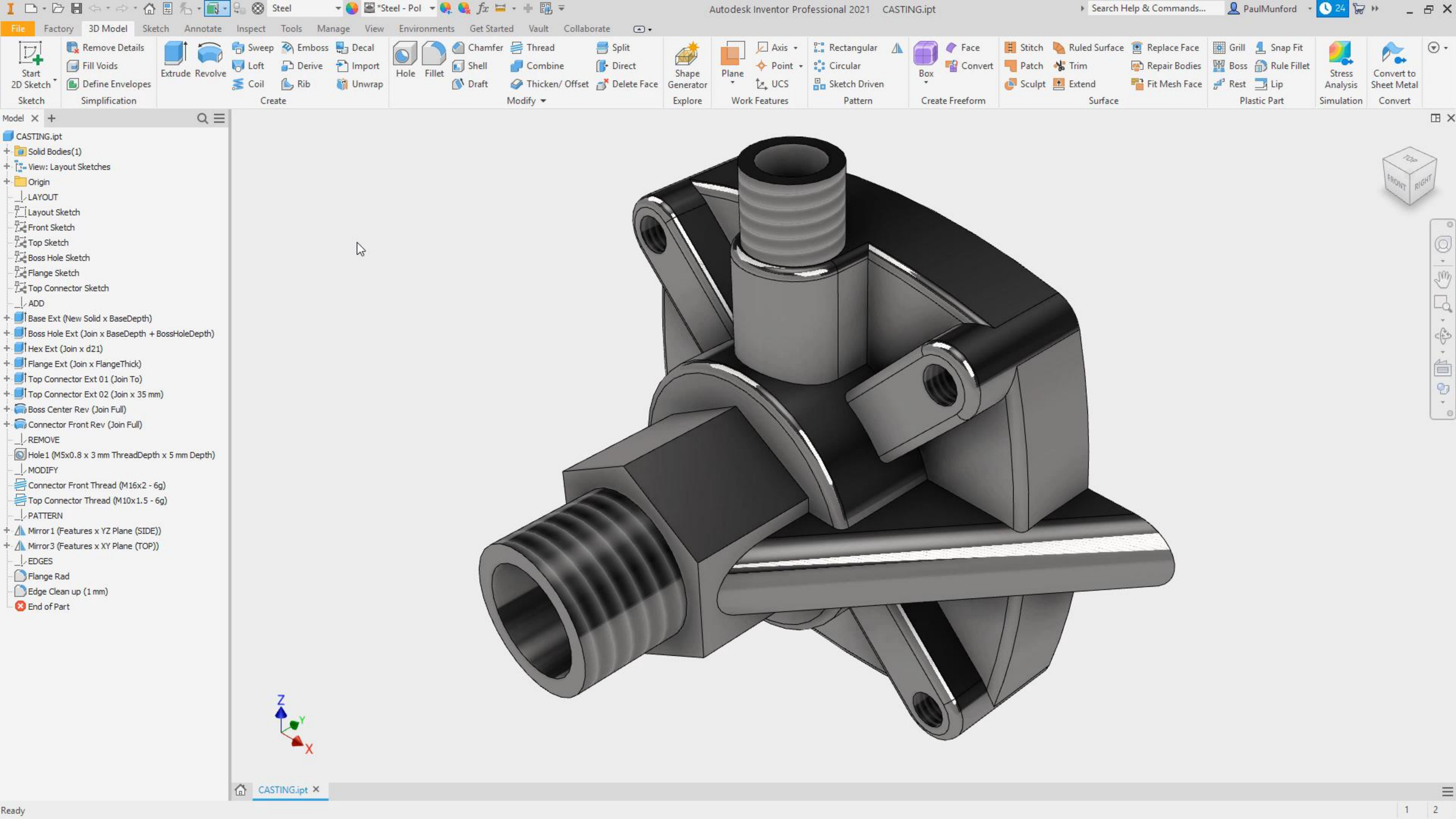
Example Feature suffixes:

Ext = Join Extrusion
Cut = Cut Extrusion
Rad = External Fillet (Radius)
Fil = Internal Fillet
Rev = Revolve
Mir = Mirror
Pat = Pattern


Parameters
Parameter Name
Consumed by
Unit/Type
Equation
Nominal Value
Drivin
Tol
Model Value
Key
Expc
Comment
Model Parameters
Reference Parameters
User Parameters
HoleDistHoriz
d0
mm
40 mm
40.000000
40.000000
Horizontal distance between holes
HoleDistVert
d2
mm
40 mm
40.000000
40.000000
Vertical distance between holes
BossDia
d10, d4, d3
mm
8 mm
8.000000
8.000000
Diameter of the Radiused bosses
BaseDepth
d14, d12, d8
mm
10 mm
10.000000
10.000000
Depth of the base feature
BossHoleDepth
d12
mm
5 mm
5.000000
5.000000
Depth of the Radiused bosses
BossCenterDepth
d14
mm
12 mm
12.000000
12.000000
Depth of the Circular feature
HexDepth
d16
mm
14 mm
14.000000
14.000000
Depth of the Hexagonal feature
FrontConnectorD...
d17
mm
12 mm
12.000000
12.000000
Depth of the front connector
TopConnectorDe...
mm
15 mm
15.000000
15.000000
Depth of the top connector
FlangeWidth
d60
mm
37.5 mm
37.500000
37.500000
Width of the triangular flange
BossCenterDia
d18
mm
30 mm
30.000000
30.000000
Diameter of the Circular feature
HexAcrossFlats
d24, d19
mm
20 mm
20.000000
20.000000
Distance across flats of the Hexagonal feature
FrontConnectorDia
d20
mm
16 mm
16.000000
16.000000
Diamter of the front connector
FlangeThick
d30
mm
8 mm
8.000000
8.000000
Thickness of the triangular flange
Add Numeric
Update
Purge Unused
Link
Immediate Update
Reset Tolerance
<< Less
Done


Autodesk Inventor Prof
File
Engineer's Notebook
Tools
View
Get Started
Vault
Vault
Collaborate
Comment
View
Arrow
Previous Note
Next Note
Select
Insert Folder
Insert Object
Finish Notebook
Notes
Format
Insert
Exit
Model
Engineer's Notebook
Thread spec' coordination
Comment 1
View 1
PAUL MUNFORD 03 September 2020
PM to check thread size of mating component before issuing part for review on 24th October 2020
Look At
Measure
Extrude
Undo
Select Other...
Work Plane
Select Tangencies
New Sketch
Repeat Parameters
3D Grips
Create Note
Assign Name...
Previous View F5
Home View F6
Properties
Help Topics...


Model
CASTING.ipt
Solid Bodies(1)
View: Layout Sketches
Master
Layout Sketches
Origin
Notes
Thread spec' coordination
LAYOUT
Layout Sketch
Front Sketch
Top Sketch
Boss Hole Sketch
Flange Sketch
Top Connector Sketch
ADD
Base Ext (New Solid x BaseDepth)
Boss Hole Ext (Join x BaseDepth
Hex Ext (Join x d21)
Flange Ext (Join x FlangeThick)
Top Connector Ext 01 (Join To)
Top Connector Ext 02 (Join x 35
Boss Center Rev (Join Full)
Connector Front Rev (Join Full)
REMOVE
Hole 1 (M5x0.8 x 3 mm ThreadDep
MODIFY
Connector Front Thread (M16x2
Top Connector Thread (M10x1.5
PATTERN
Mirror1 (Features x YZ Plane (SIDI
Mirror3 (Features x XY Plane (TOF
EDGES
Flange Rad
Edge Clean up (1 mm)
End of Part
PAUL MUNFORD 03 September 2020
PM to check thread size of mating component before issuing part for review on 24th October 2020

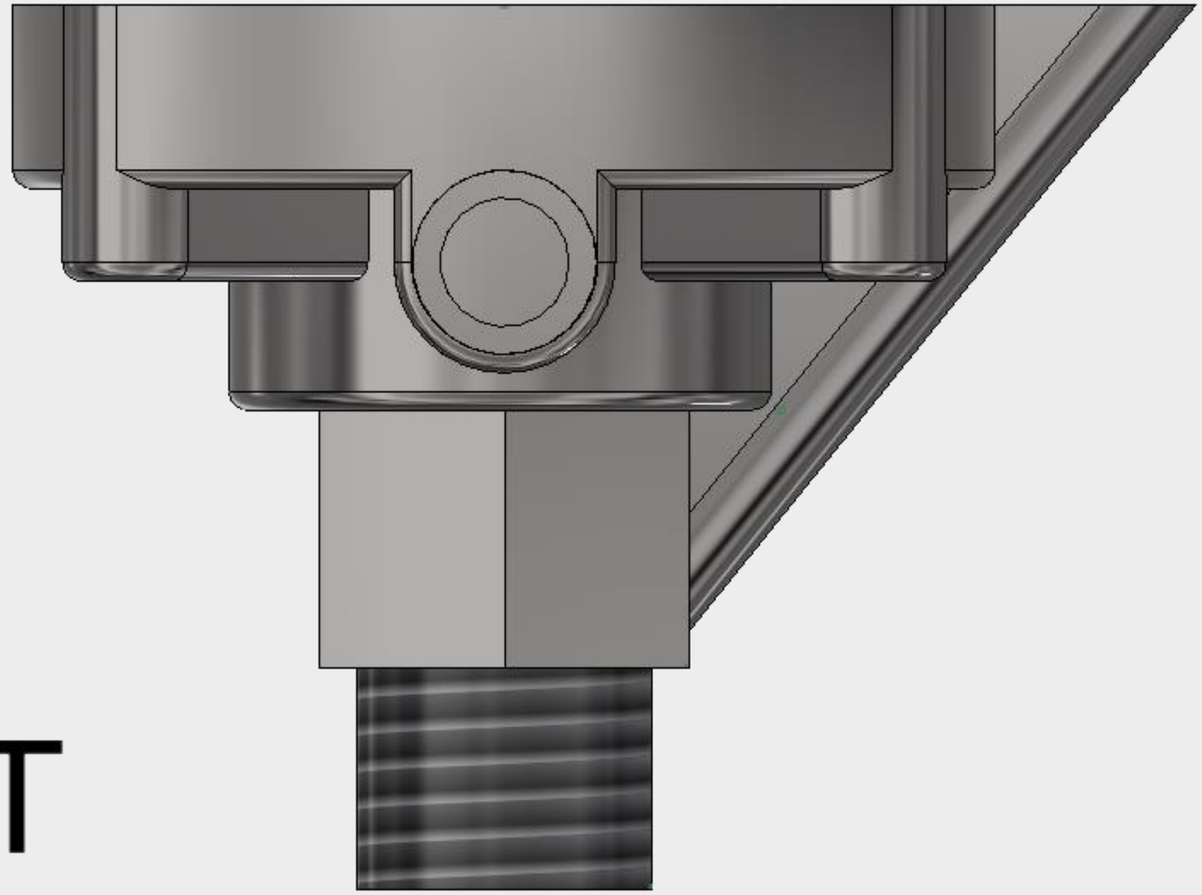

Autodesk Inventor Professional 2021 CASTING.ipt
Search Help & Commands...
PaulMunford
File Factory 3D Model Sketch Annotate Inspect Tools Manage View Environments Get Started Vault Collaborate
Start 2D Sketch
Remove Details
Fill Voids
Define Envelopes
Extrude Revolve
Sweep Loft Coil
Emboss Derive Rib
Decal Import Unwrap
Hole Fillet
Chamfer Shell Draft
Thread Combine Thicken/ Offset
Split Direct Delete Face
Shape Generator
Plane
Axis Point UCS
Rectangular Circular Sketch Driven
Box
Face Convert
Stitch Patch Sculpt
Ruled Surface Trim Extend
Replace Face Repair Bodies Fit Mesh Face
Grill Boss Rest
Snap Fit Rule Fillet Lip
Stress Analysis
Convert to Sheet Metal
Sketch Simplification Create Modify Explore Work Features Pattern Create Freeform Surface Plastic Part Simulation Convert
Model
CASTING.ipt
Solid Bodies(1)
View: Layout Sketches
Origin
LAYOUT
Layout Sketch
Front Sketch
Top Sketch
Boss Hole Sketch
Flange Sketch
Top Connector Sketch
ADD
Base Ext (New Solid x BaseDepth)
Boss Hole Ext (Join x BaseDepth + BossHoleDepth)
Hex Ext (Join x d21)
Flange Ext (Join x FlangeThick)
Top Connector Ext 01 (Join To)
Top Connector Ext 02 (Join x 35 mm)
Boss Center Rev (Join Full)
Connector Front Rev (Join Full)
REMOVE
Hole1 (M5x0.8 x 3 mm ThreadDepth x 5 mm Depth)
MODIFY
Connector Front Thread (M16x2 - 6g)
Top Connector Thread (M10x1.5 - 6g)
PATTERN
Mirror1 (Features x YZ Plane (SIDE))
Mirror3 (Features x XY Plane (TOP))
EDGES
Flange Rad
Edge Clean up (1 mm)
End of Part
TOP FRONT RIGHT
Z Y X
CASTING.ipt
Ready

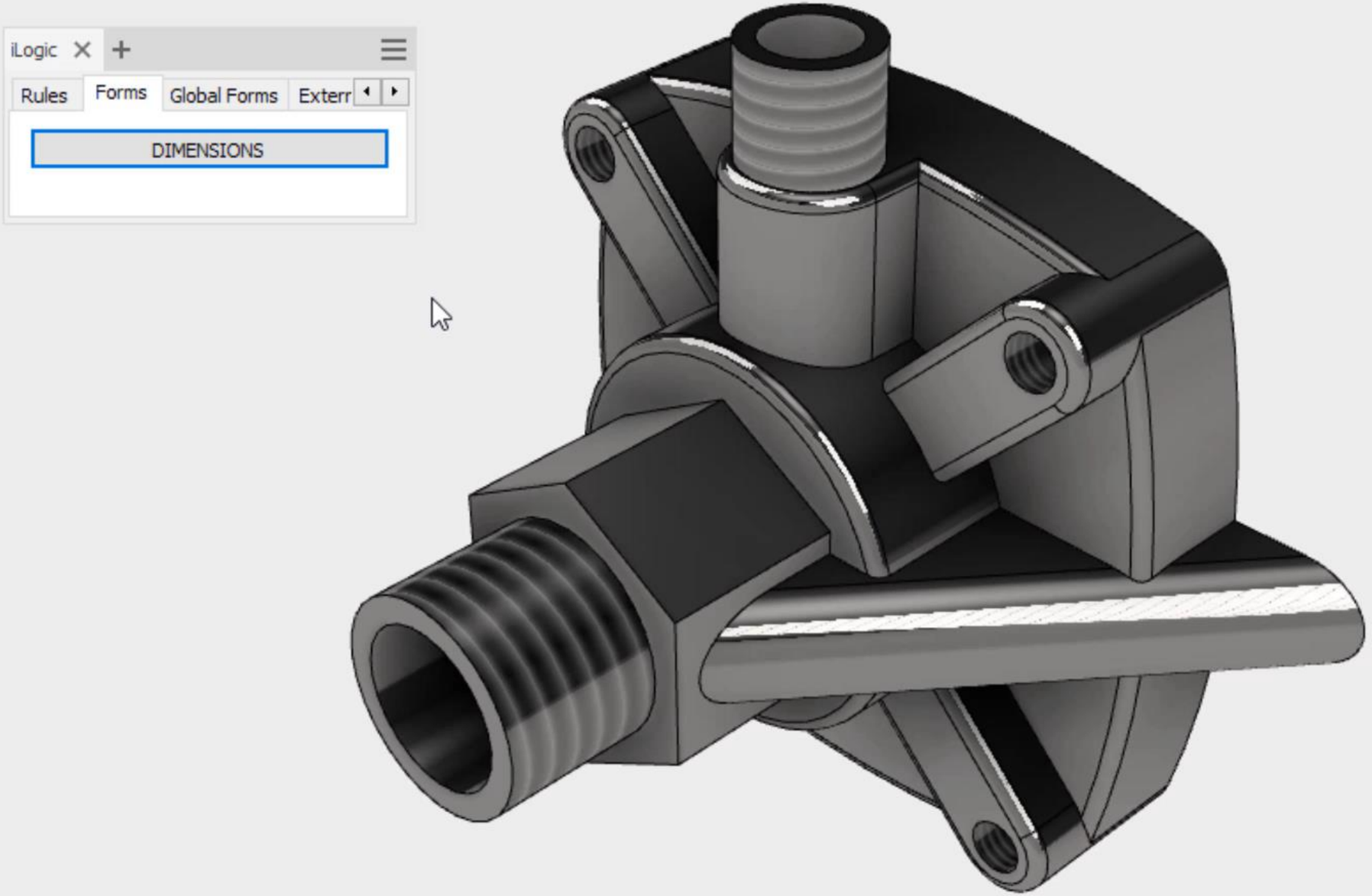
- CASTING.ipt
  - Solid Bodies(1)
  - View: Master
  - Origin
  - LAYOUT
  - Layout Sketch
  - Front Sketch
  - Top Sketch
  - Boss Hole Sketch
  - Flange Sketch
  - Top Connector Sketch
  - ADD
  - Base Ext (New Solid x BaseDepth)
  - Boss Hole Ext (Join x BaseDepth + BossHoleDepth)
  - Hex Ext (Join x d21)
  - Flange Ext (Join x FlangeThick)
  - Top Connector Ext 01 (Join To)
  - Top Connector Ext 02 (Join x 35 mm)
  - Boss Center Rev (Join Full)
  - Connector Front Rev (Join Full)
  - REMOVE
  - Hole1 (M5x0.8 x 3 mm ThreadDepth x 5 mm Depth)
  - MODIFY
  - Connector Front Thread (M16x2 - 6g)
  - Top Connector Thread (M10x1.5 - 6g)
  - PATTERN
  - Mirror1 (Features x YZ Plane (SIDE))
  - Mirror3 (Features x XY Plane (TOP))
  - EDGES
  - Flange Rad
  - Edge Clean up (1 mm)
  - End of Part

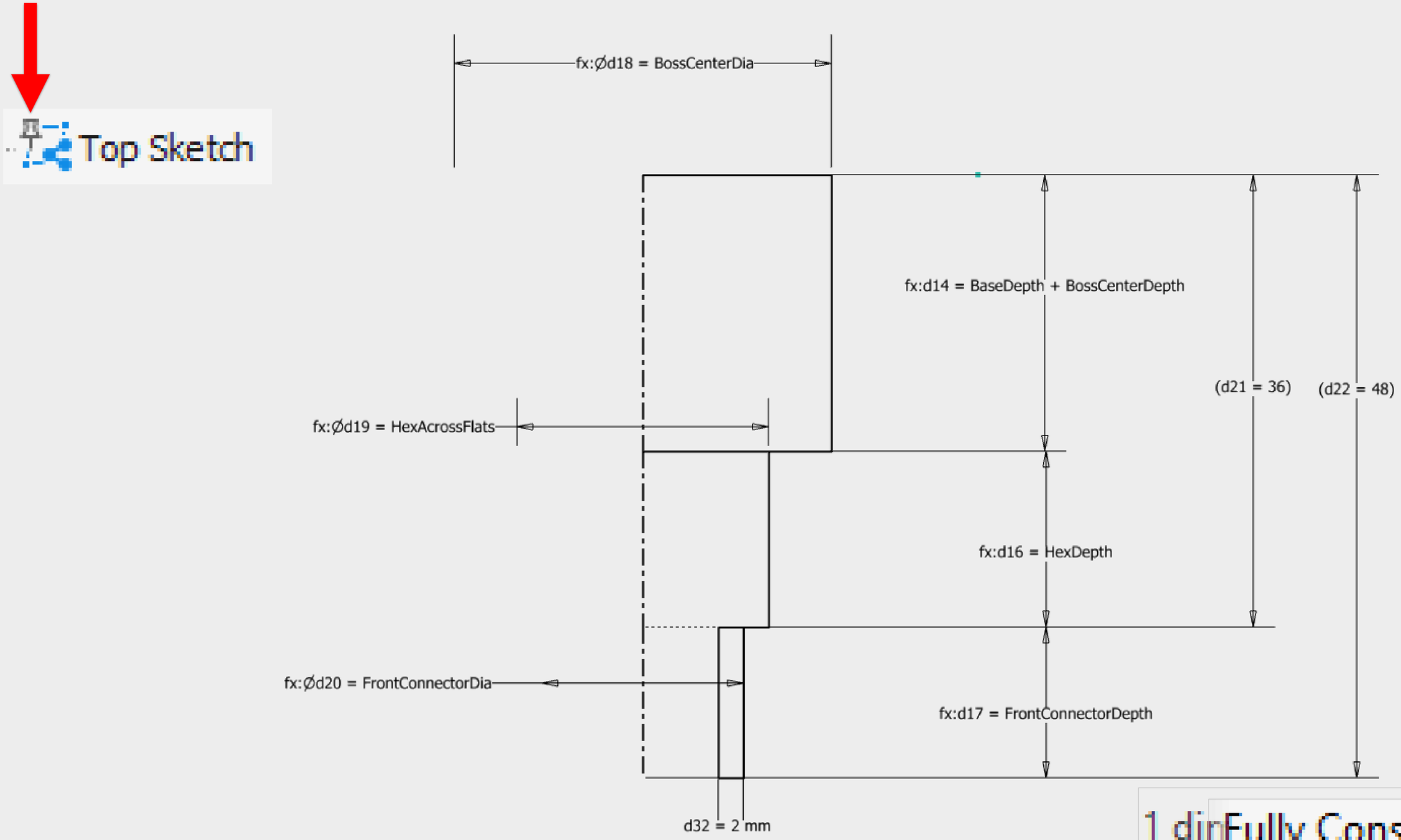
BOSS

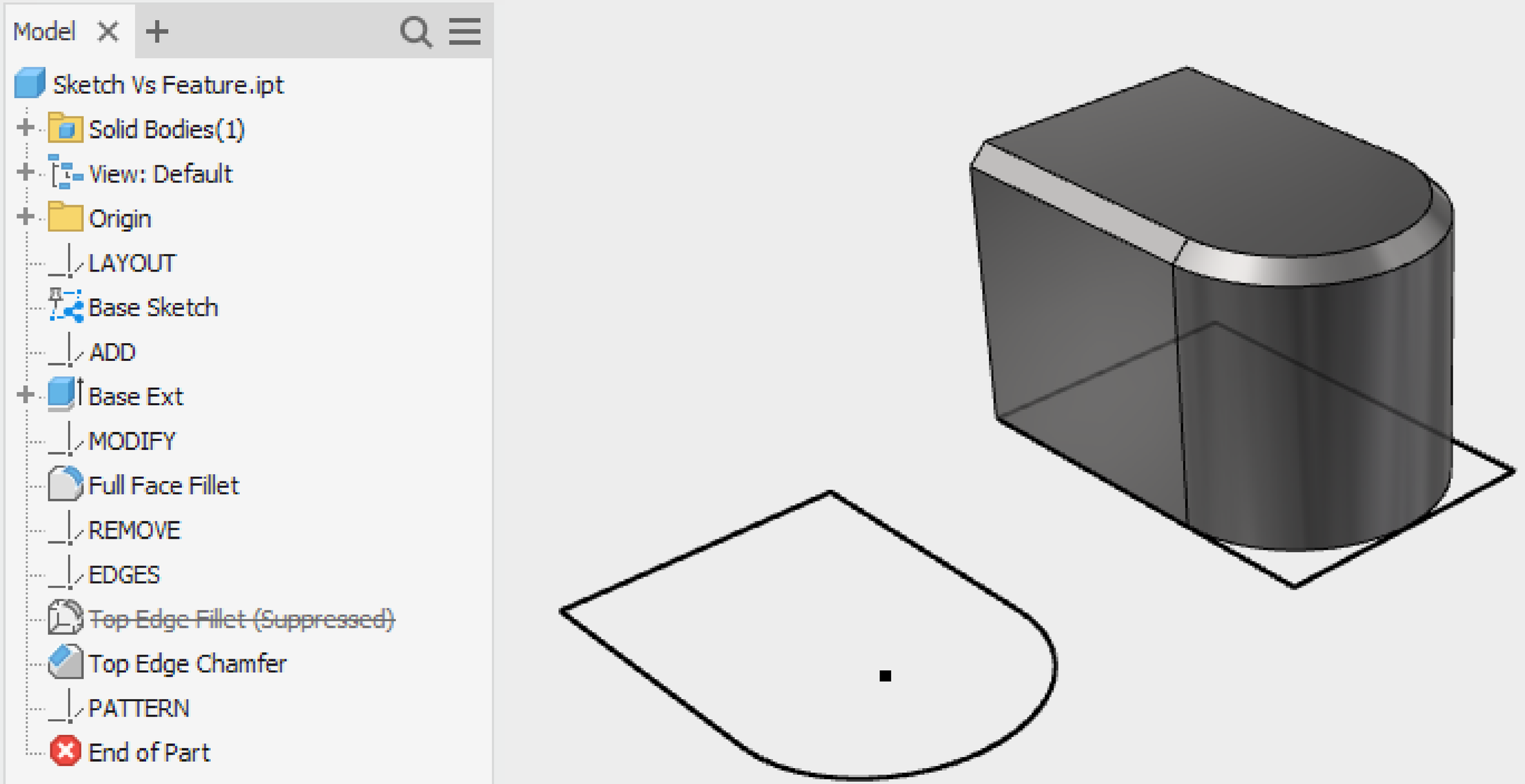
HEX

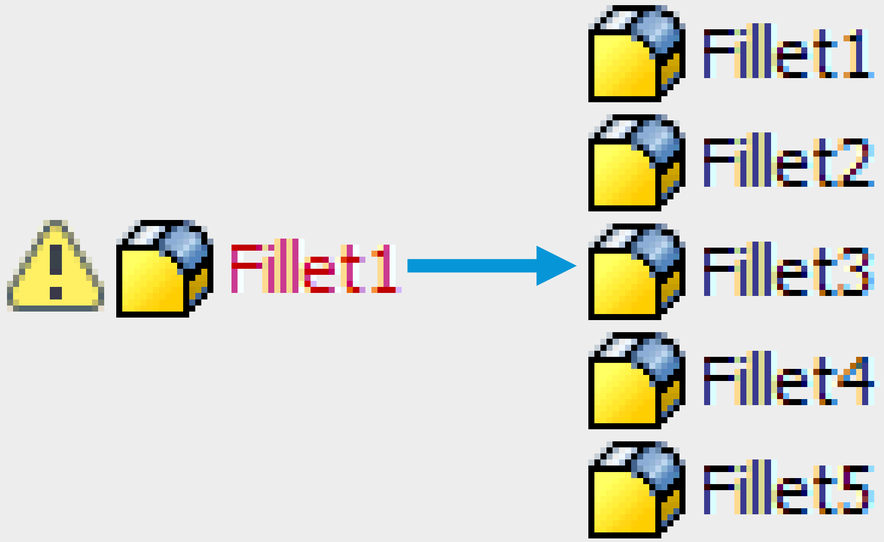
FRONT CONNECTOR

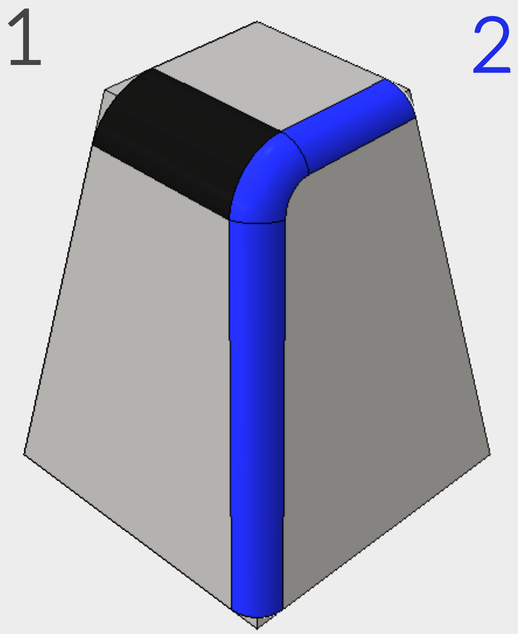


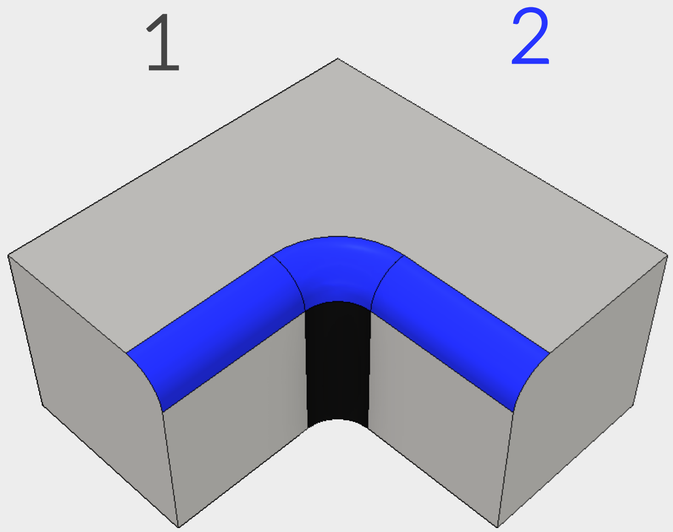

iLogic
Rules
Forms
Global Forms
Exterr
DIMENSIONS


Top Sketch
fx:Ød18 = BossCenterDia
fx:d14 = BaseDepth + BossCenterDepth
(d21 = 36)
(d22 = 48)
fx:Ød19 = HexAcrossFlats
fx:d16 = HexDepth
fx:Ød20 = FrontConnectorDia
fx:d17 = FrontConnectorDepth
d32 = 2 mm
1 din
Fully Constrained


Model
Sketch Vs Feature.ipt
Solid Bodies(1)
View: Default
Origin
LAYOUT
Base Sketch
ADD
Base Ext
MODIFY
Full Face Fillet
REMOVE
EDGES
Top Edge Fillet (Suppressed)
Top Edge Chamfer
PATTERN
End of Part


Fillet1
Fillet1
Fillet2
Fillet3
Fillet4
Fillet5


1
2


1
2

Reliable Modelling Techniques for Complex Part Design in Inventor

Summary

- Before you start – stop! Make a plan
  (use the checklist from the handout).
- Standardize the Application settings and Your Templates.
- Take charge of your Relationships.
- Flex. Don’t leave booby traps to trip you up later!
- Document your design intent.
- Roll back the EOP, and look for ways to improve for next time.

Q&A (My Turn!)

Q: Name Paul's two criteria for a well modelled part?

A: Correct Geometry and Easy to update.

Q: Why do we fully constrain sketches?

A: 'Fully constrained means fully predictable!' (Paul Said so).

Q: List Paul's four relationship rules.

A: Minimum, Intended, Planed, Obvious.

Q: List Paul's Relationships - order of preference.

A: Parametric, Sketch to Sketch, Sketch to Feature, Feature to Feature.

Q: Why do edge consuming features come last?

A: Creating a relationship with a consumed edge can cause instability.

Please help me by recommending this class!

@PaulCADMunford

Q&A

Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2020 Autodesk. All rights reserved.