Model to machine in 90s: smart, fast templating for fusion 360 CAD-CAM workflows

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About the speaker

INDUSTRIAL DESIGNER & MACHININST

Chris Chekan oversees the workshop at the Autodesk Technology Center - Toronto, where he helps internal and resident teams alike to push the boundaries of design-make workflows. He looks forward to a day when the necessity of translating between “Designers” and “Manufacturers” is a relic of a dark past.

AUTODESK TECHNOLOGY CENTERS

The residency program at the Autodesk Technology Centers provides open workspaces and equipment for teams from industry, academic, and startup communities doing forward-looking work in the areas of construction, manufacturing, and emerging technologies.

OVERVIEW: **CAD** *and* **CAM**

WHAT MAKES FUSION360 SO WELL SUITED TO RAPID PROTOTYPING, PRODUCT DEVELOPMENT AND LEAN MANUFACTURING?

SPEED + FLEXIBILITY

Demonstrate how to leverage the inherit strengths of Fusion360's parametric modelling environment to produce manufacturing template files that can expedite programming new parts and automate repetitive tasks in CAD-CAM workflows.

With CAD and CAM under the same roof, you can create robust template files that adapt to changes in **tooling**, **stock sizes**, **work holding**, **machines** and even the **parts themselves** without ever having to re-design or re-program.

Why even bother with manufacturing template files?

- no one, ever

REPETITION KILLS PRODUCTIVITY

CAM programming is filled with repetitive tasks that kill productivity. Automate and standardize your way out of needless mouse clicks!

DESIGNERS HAVE DONE IT LONGER

Like it or not, CAD designers have long ago figured out clever workarounds to streamline and automate their design workflows. With Fusion360, you can do the same with CAM.

Why even bother with manufacturing template files?

HUMAN ERROR IS UNAVOIDABLE

Regardless of the experience level, attention to detail or other genius of the individual, at the end of the day we're all human – and to be human is to make mistakes.

STANDARDIZE MANUFACTURING FILES

Master Template files create a level of standardization across a team that helps with group resiliency and knowledge sharing.

LEAN IN TO LEAN

Only one machine available? But part was programmed for another? **No problem**.

Out of standard stock sizes? **No problem**.

Programmed for tooling that hasn't arrived yet? **No problem**.

- no one, ever

MAKE BETTER PARTS

Dialed-in tool paths make for better parts. If you used to spend an hour programming a part, but now have the base template done in 90s, just think of all of the fine tuning and optimization…

WHAT MAKES A GOOD MANUFACTURING TEMPLATE?

MACHINE SPECIFIC

Create a Master Template file for each machine on the shop floor, or alternatively for each machine configuration.

WORK HOLDING / FIXTURES

Correctly modelled work holding and fixtures allow for accurate simulation, operation patterning, stock sizing and part positioning, all of which can be easily updated as new parts come in, or designs are optimized/refined.

PARAMETRIC STOCK MODELS

Don’t have standard stock sizes? No problem – program your part with generic stock, and update the final as-cut dimensions using User Defined Parameters to update vise positioning and toolpaths based on final dimensions.

GENERIC REFERENCE GEOMETRY

Surfacing or engraving with Ball end mills? Need to tilt to avoid collisions from the spindle going deep into a pocket? Having generic reference geometry available in the Master Template makes this a quick, no brainer.

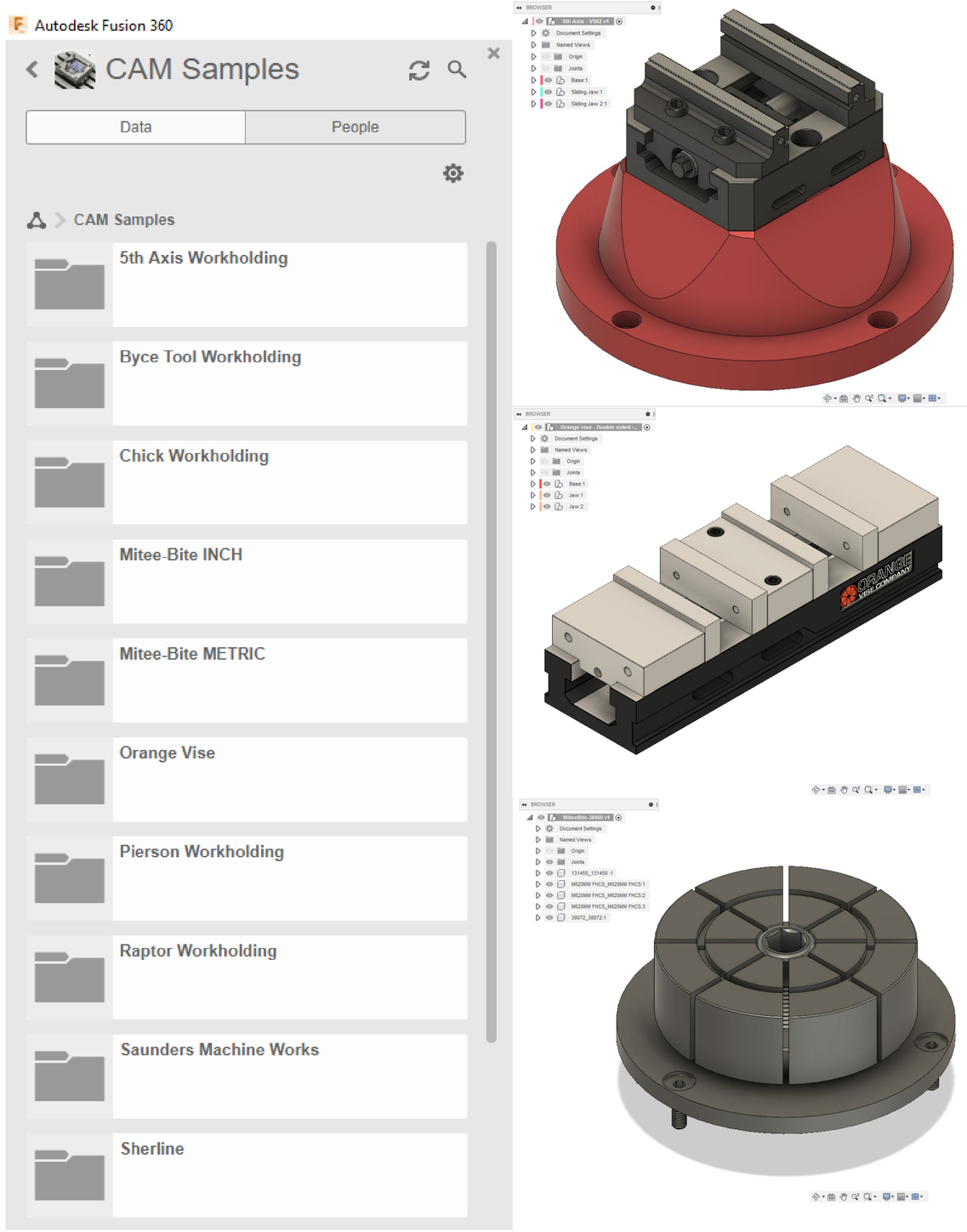
WORK HOLDING IN FUSION 360

Library of work holding models already lives in Fusion

- From main data panel, scroll down to CAM SAMPLES
  - Locate the MFG folder (or Generic)
    - Once opened, saving a copy will allow for edits and customization

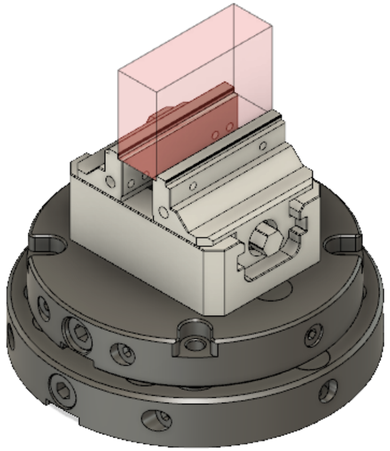
Easily import .STEP/.IGES models from MFG websites

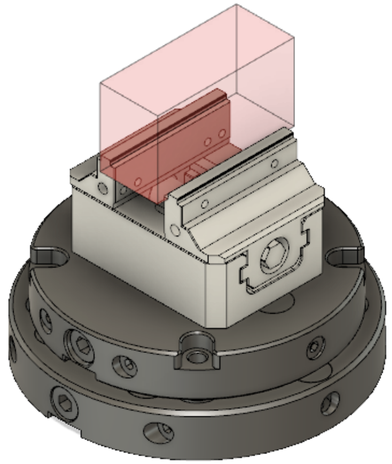
- Download geometry directly from MFG website
  - In your own project, select the UPLOAD button
    - define join relationships, save as template

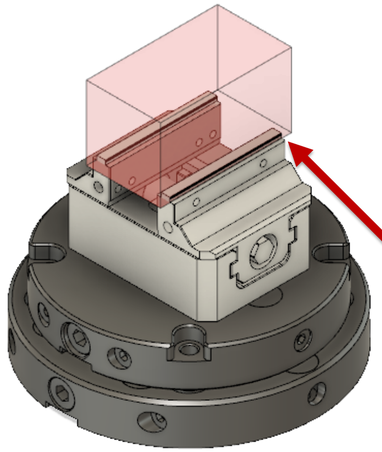


PARAMETRIC STOCK MODEL

If design is parametrically sound, stock dimensions, part position and vise jaw positioning will update automatically throughout final posted program.







Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameter	StockLength	mm	76.2 mm	76.20	
User Parameter	StockWidth	mm	25.4 mm	25.40	
User Parameter	StockHeight	mm	50.8 mm	50.80	
User Parameters					
Model Parameters					

OK

Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameter	StockLength	mm	90 mm	90.00	
User Parameter	StockWidth	mm	38.1 mm	38.10	
User Parameter	StockHeight	mm	50.8 mm	50.80	
User Parameters					
Model Parameters					

OK

Parameters

Parameter	Name	Unit	Expression	Value	Comments
Favorites					
User Parameter	StockLength	mm	90 mm	90.00	
User Parameter	StockWidth	mm	50.8 mm	50.80	
User Parameter	StockHeight	mm	50.8 mm	50.80	
User Parameters					
Model Parameters					

OK

Warning

LeftJaw_narrow-stock (
Compute Failed
RightJaw_narrow-stock (
Compute Failed
Some positions cannot be resolved due to joint conflicts

Close

What does *Parametric Modelling* actually mean?

- me, far too recently

PARAMETRIC MODELLING BASICS

Parametric modeling is a way to record your design as a series of steps or actions – like a recipe for your model. This can be re-wound, modified and replayed at any time, allowing for design changes to update every related downstream operation leading to the completed model.

It is critically important to structure your design correctly in order to avoid errors in updating your design based on parameter

TIMELINE

The timeline is a graphical representation of your parametric design history. It is categorized by component/assembly and can be colorized in order to easily track related operations.

Groups can be created to better organize and locate individual operations.

PARAMETRIC MODELLING BASICS – CONT'D

NAME EVERYTHING

This is a good habit to get into form the get go and will make your life exponentially easier when updating/editing parametric designs.

ASSEMBLY OPERATIONS

Unique to Fusion, any time a new component or assembly is created, it is logged as a timeline event. Similarly the ability to "capture position" of components within the design history allows for position-dependent modeling.

GOLDEN RULE: ALWAYS BE ACTIVATING

Components carry their own dedicated timeline, making it critical to ***activate*** components before editing anything related to them, in order to capture the design history accurately.

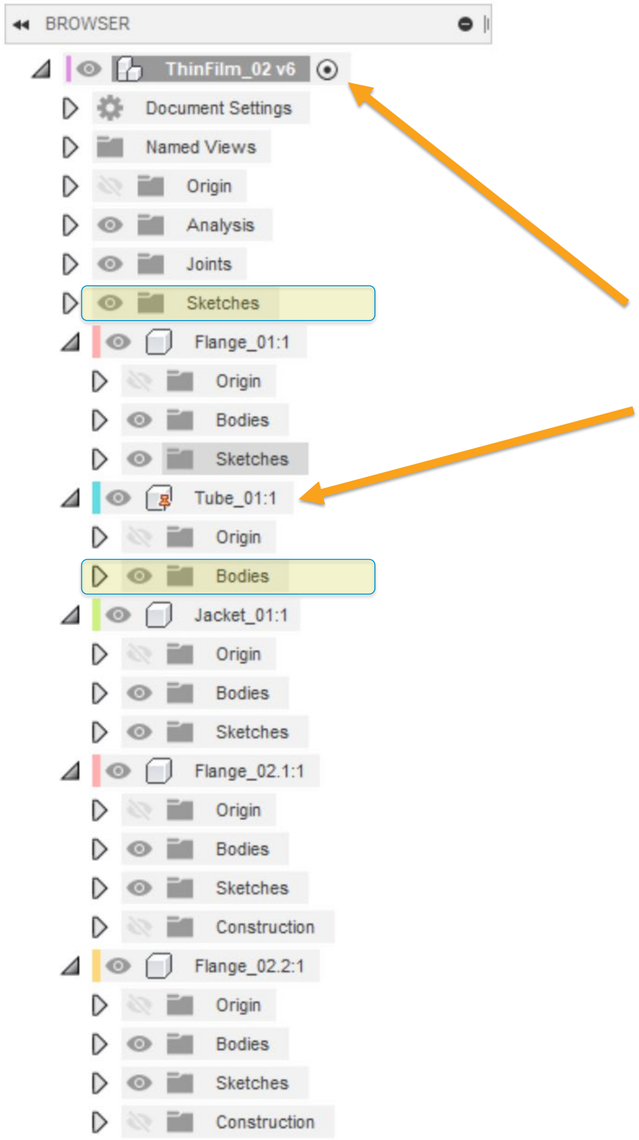
PARAMETRIC MODELLING BASICS – CONT'D

BROWSER TREE STRUCTURE

Think of this as a design "order of operations" – components should be created before sketches, and bodies are created within them.

In order to do so, GOLDEN RULE must be followed: Activate each component before creating sketches, bodies or construction geometry within it.

Base or Skeleton sketches are often created in the base Assembly, but should be simple and used sparingly to avoid any confusion in the history.



The "Tube_01:1" component does not contain any sketches, but it does contain a body. This was created using the base design sketch, located in the root of "ThinFilm_02" assembly.

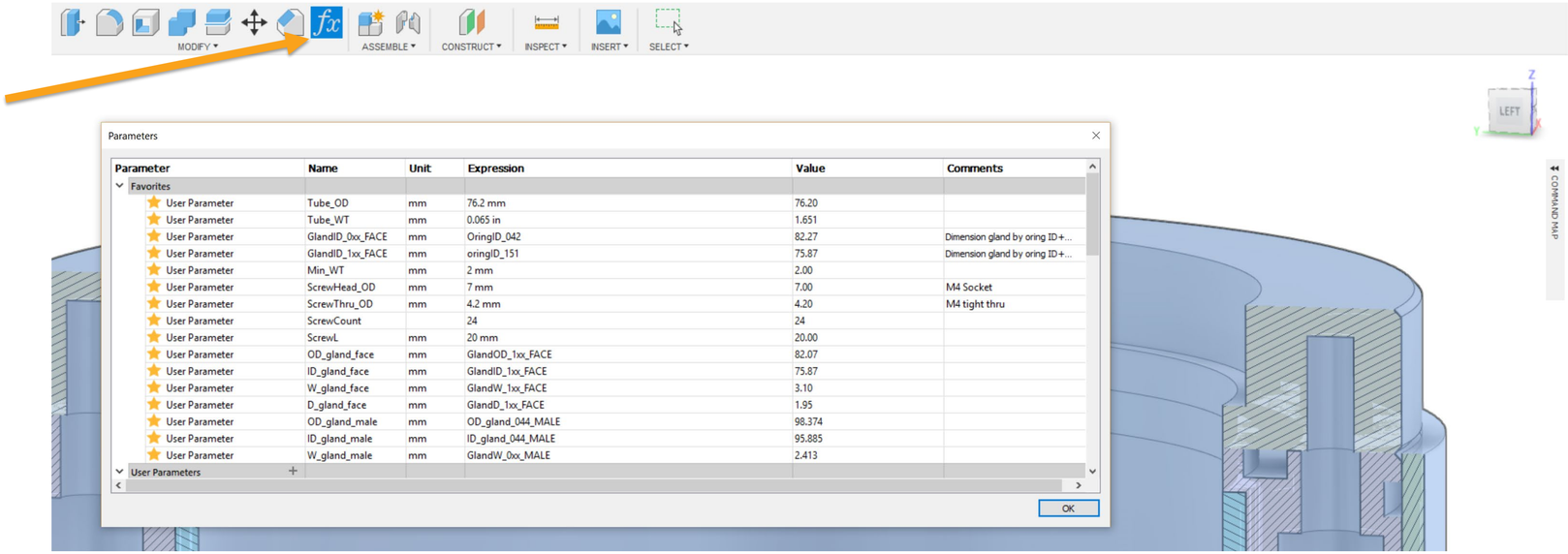
PARAMETRIC MODELLING BASICS – CONT'D

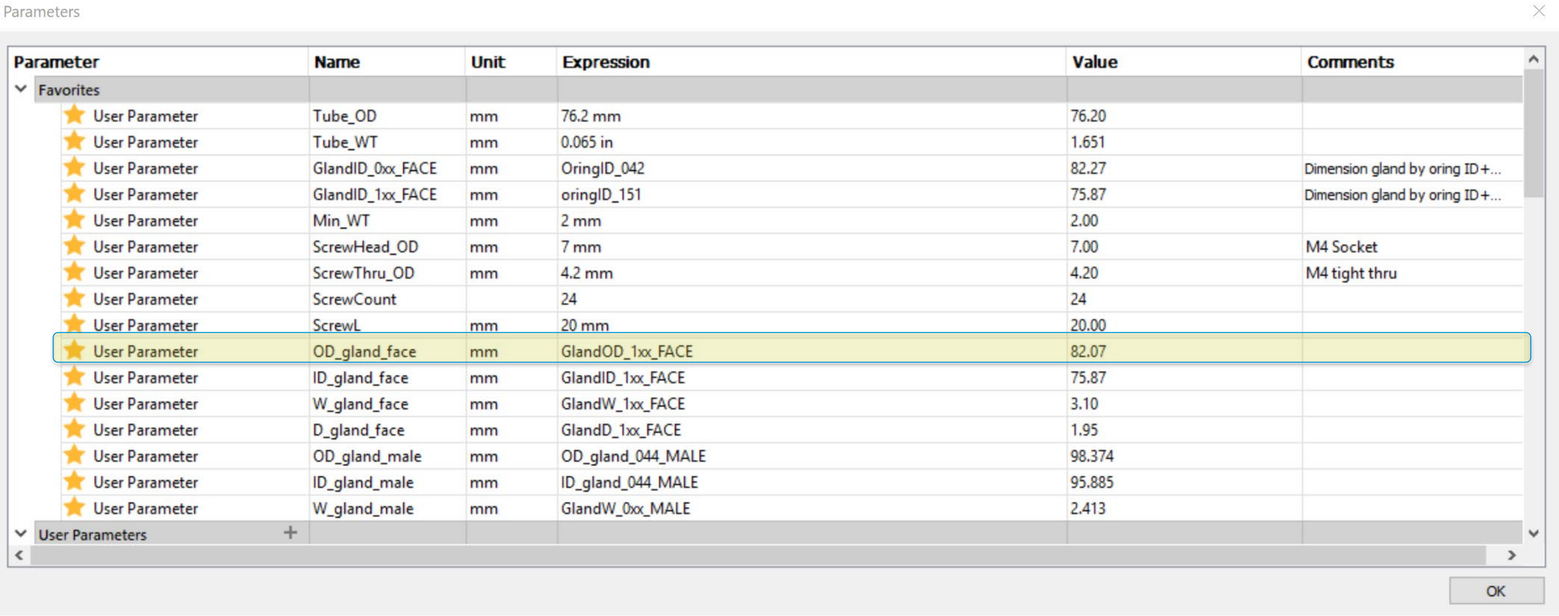
USER DEFINED PARAMETERS

Anytime you have the option of defining a parameter in Fusion, you can enter either: a numerical value, an existing dimension, enter a function or select a User Defined Parameter.

User Defined Parameters can themselves be functions or reference other parameters, have dedicated units or none at all (pattern count for example).

These are powerful ways of building robust and easily configurable designs.

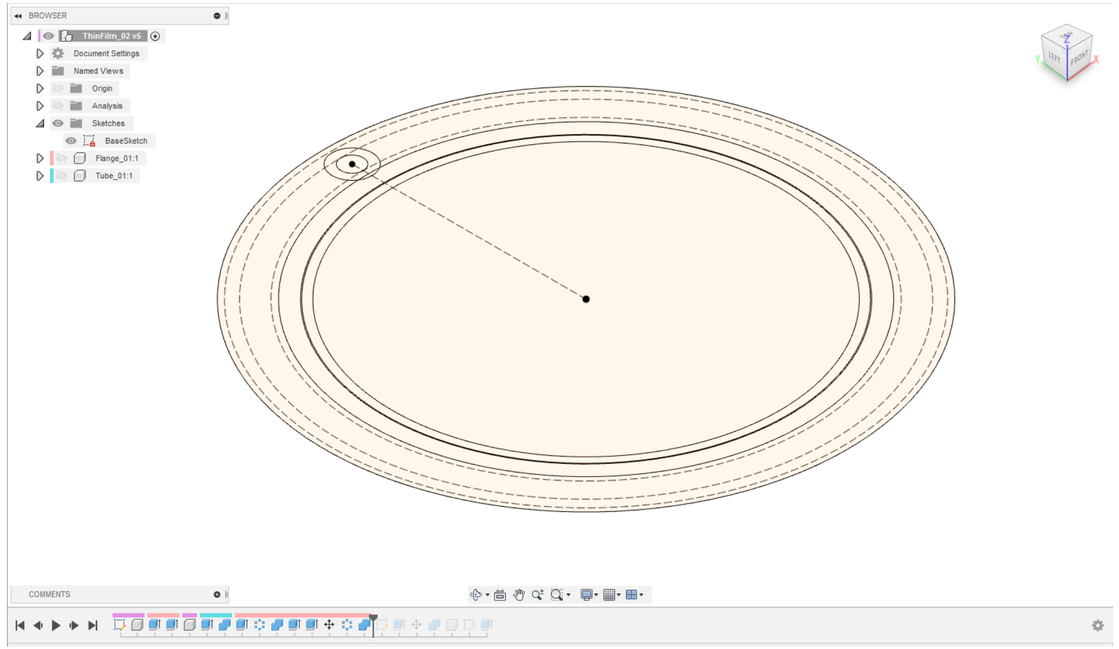


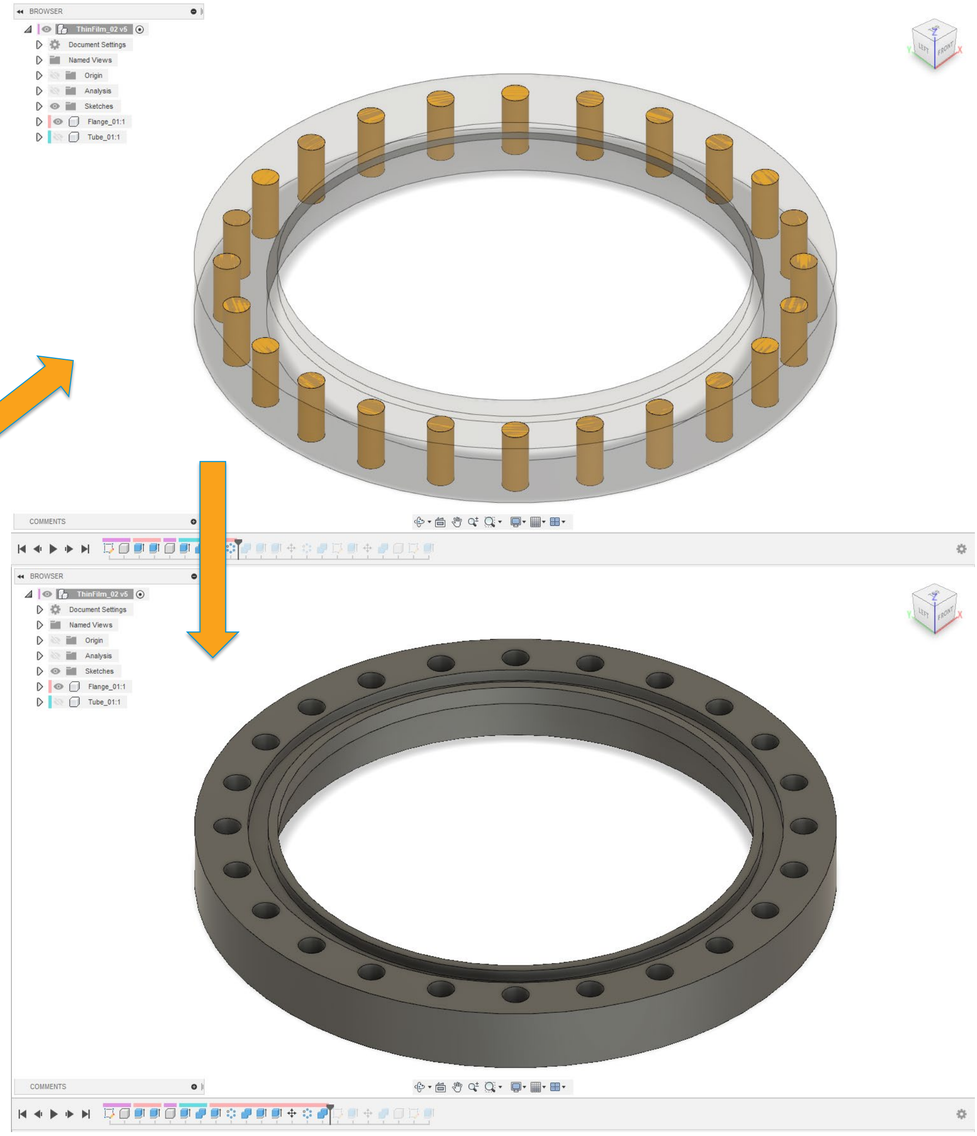


PARAMETRIC MODELLING BASICS – CONT'D

PATTERN/MODIFY FEATURES, NOT SKETCHES

Keep sketches as simple as possible, and any patterns, combinations, stamps or other modifications should happen with the extruded bodies rather than the sketch itself.



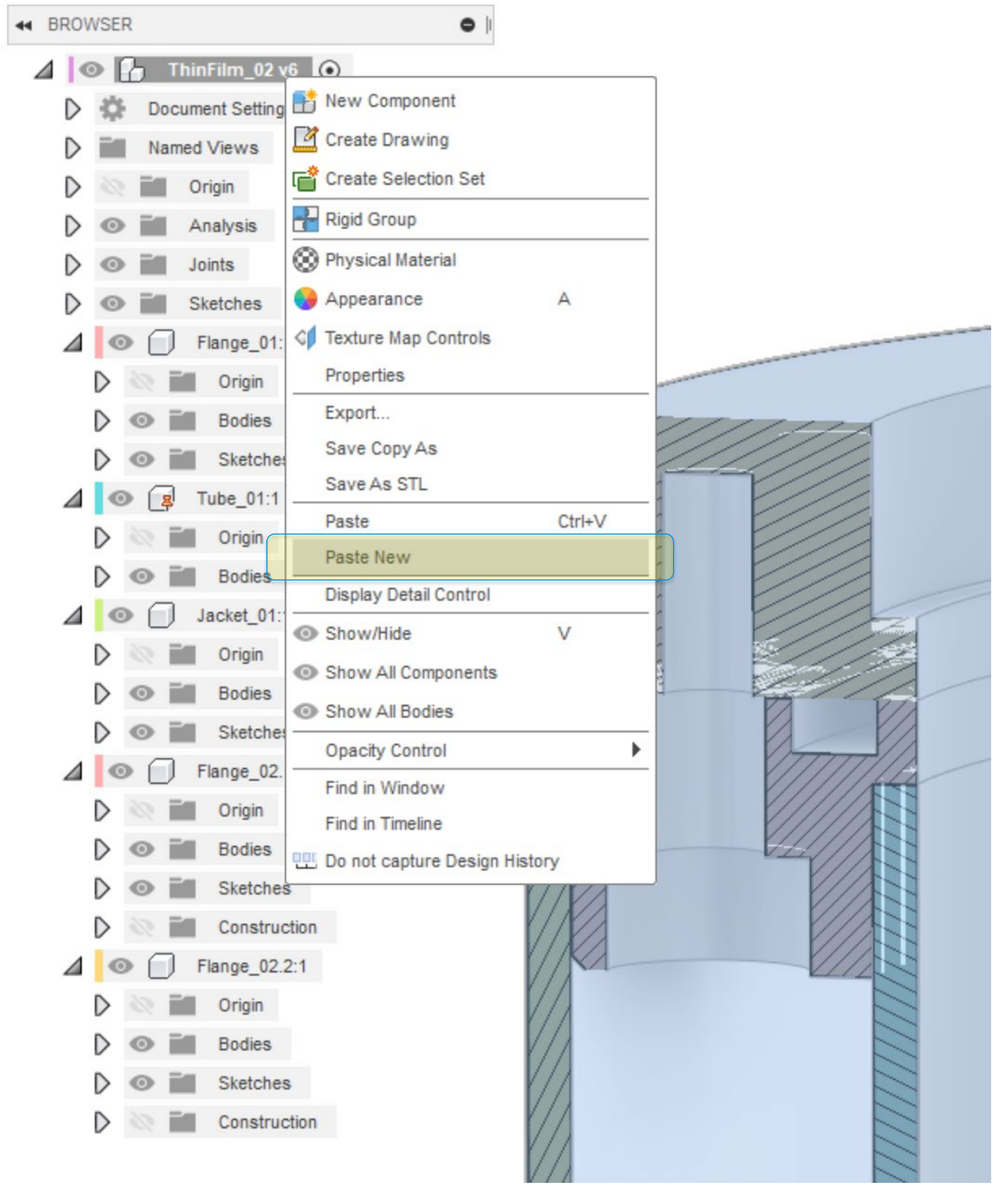


PARAMETRIC MODELLING BASICS – CONT'D

COPY > PASTE VS. PASTE NEW

Bodies are linked to their sketches and reference components **always** – copy and paste a body will keep same design history for the new body as the original.

This is optional when copying ***Components***: **Paste New** creates a new component with duplicated, but not linked design history to the original. It's as if you modelled them both separately, from the same references. This is powerful for creating diverging design options, or exploratory iterations of a part.



PARAMETRIC MODELLING BASICS – CONT'D

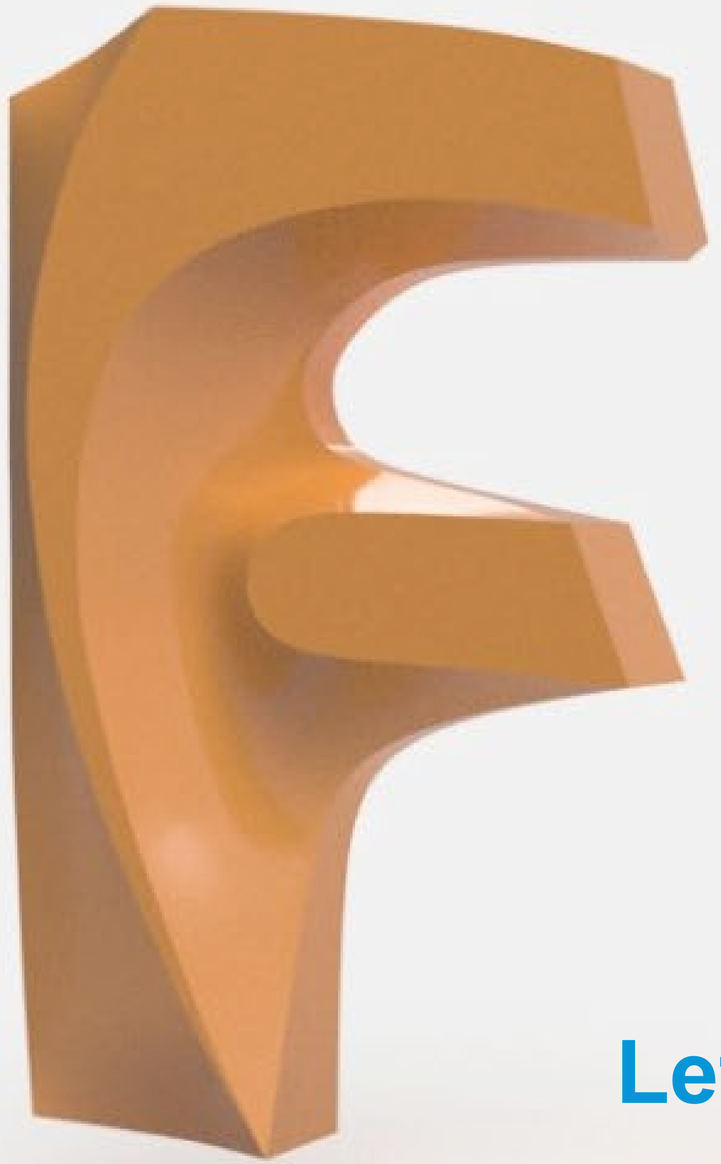
HEIRARCHY OF REFERENCE RELATIONSHIPS

Not all references are created equally!

There is a hierarchy of "stability" when it comes to referencing existing geometry. It is best practice to reference the most stable feature/body/geometry when creating new features to avoid calculation errors downstream

OVER-GENERALIZED ORDER OF STABILITY:

1. ORIGIN, "WORK GEOMETRY" MOST STABLE
   - Planes, Axes created from design origin
   - Planes, Axes created from Brep references are less stable
2. BODY REFERENCES ARE MORE STABLE THAN FACES
   - If WORK GEOMETRY is unavailable to reference, bodies are preferable to EDGE/FACE references
3. FACE REFERENCES ARE MORE STABLE THAN EDGES
   - If BODY references are unavailable, FACES are more stable than EDGES
     - EDGES are intersection of two faces
4. EDGE REFERENCES ARE MORE STABLE THAN POINTS
   - If FACE references are unavailable, EDGES are more stable than POINTS



Let’s see some of this theory in action…

Let’s keep the conversation going

Connect with the Technology Center team and residents at the Community Zone Pods daily from 1 – 2pm!

autodesktechnologycenters.com/apply-now/

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