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Ins and Outs of Using Adaptivity in Autodesk Inventor®

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MA34-1 Learn when -- and when not -- to use Autodesk Inventor's Adaptive technology in sketches, features, parts, and assemblies. This class will give an in-depth look into the different types of adaptivity, focusing on the strengths and weaknesses of each option.

About the Speaker:

Keith has been the senior application engineer for D3 Technical Services in Springfield, Missouri for the last 7 years. He holds a B.S. degree in Drafting and Design Technology from Southwest Missouri State, and has been an instructor at AU the past 2 years. He is both an Autodesk Inventor Certified Expert and an Autodesk Manufacturing Solutions Implementation Certified Expert. Keith's main focus at D3 is implementing Autodesk Inventor, Vault, and Productstream with new customers, as well as 2D/3D CAD automation/configurator development for existing customers. He also works as an engineering design consultant.

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What is Adaptivity?

Autodesk's Definition:

Adaptive geometry can be sized and positioned in the context where it is used. When you designate underconstrained geometry as adaptive, you specify the geometric elements allowed to change, while controlling the elements that you want to remain a fixed size or position.

What does this mean?

Adaptivity is functionality, within Inventor, that allows the size of a part/feature to be determined by setting a relationship to another part in an assembly. Basically, adaptivity is a special way to add constraints. These constraints differ from regular constraints in that they are driven from a separate file. This separate file can be an assembly file or another part within the assembly file. A good example of adaptivity is constraining a shaft to a hole in another part. If set up correctly, when the size of the hole changes the diameter of the shaft updates as well.

In order to fully understand adaptivity one must have a good understanding of the normal (non-adaptive) constraints inside Inventor. Without this understanding, adaptivity will cause parts and assemblies to react in unimaginable ways and cause intense frustration.

When Should Adaptivity Be Used?

Adaptivity is normally used during the initial design phase of a model, when changes are made rapidly and many parts are affected. Once a design is released, and parts become standard parts available for use in other designs, adaptivity should be removed to eliminate the possibility of inadvertently changing a released design. Removing adaptivity also improves performance.

As with using any other constraint in Inventor, forethought should be given to how a design may change before adaptivity is applied. If a part is not likely to change, it is better to apply normal (non-adaptive) constraints. Adaptivity should be used only when absolutely necessary.

Types of Adaptivity

There are a number of different ways to add adaptive constraints and it is important to look at each one separately in order to fully understand the power of adaptivity. While each of these fall under the category of adaptivity they all react somewhat differently.

Adaptive Sketches – Cross Part Sketch Geometry

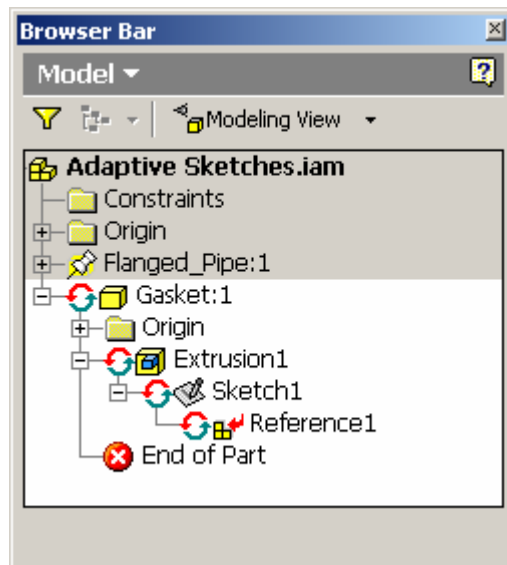
The first type of adaptivity is adaptive sketches using cross part sketch geometry. When editing a part in the context of an assembly it is possible to use the Project Geometry command to copy edges from other parts in the assembly into the current sketch. This is very helpful when designing parts with mating features.

In order for the projected geometry to be adaptive to the original part the correct setting must be set in the Application Options – Assembly Tab.

Cross Part Geometry Projection

Enable Associative Edge/Loop Geometry Projection During In-Place Modeling

With this option checked, edges projected from one part into a sketch in another part are automatically added as a Reference object. An icon for this object appears in the browser under the sketch icon (see image below). The sketch, feature, and part are also automatically made adaptive.



If the option is turned off the edges will still be projected. However, they will not be linked back to the original part and a Reference object will not be created. The geometry will appear as a different color in the graphics window. In order to dimension or change this geometry you will need to right-click on it and select Break Link. This will cause it to become regular sketch geometry.

When projecting geometry it is possible to toggle the option by holding the Control key. For example, if the option is checked, holding the Control key causes the projected edges to not be linked.

It is also important to note that when the option is checked an entire face can be projected at once rather than selecting each entity separately. This method groups all the entities together as one

Reference object. This multiple entity Reference object will react differently to changes than single entity Reference objects.

The adaptivity icon (the curved arrows next to the part name in the browser) denotes the part as being adaptive. Notice this icon is also next to the Feature, Sketch, and Reference icons. Each level must be made adaptive in order for the part to update correctly. This gives the user better control over which features may have adaptive constraints applied.

To temporarily remove the adaptive constraint from the sketch, right-click on the Reference icon in the browser and uncheck the Adaptive option in the context menu. This will prevent the sketch and associated features from updating when the part it was projected from changes. If adaptivity is turned back on and updated, the sketch will update to meet the changes of the other part.

To permanently remove the adaptive constraint, right-click on the Reference icon in the browser and select Break Link from the context menu. Once the link is broken it cannot be restored.

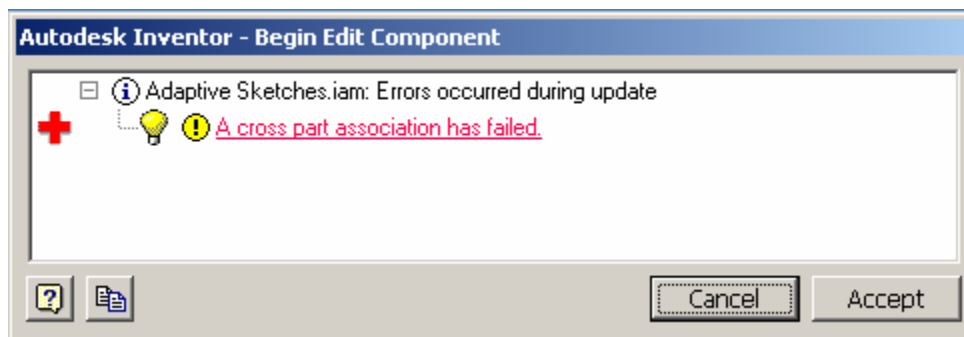
Since the projected geometry is associated to another part in the assembly some degrees of freedom are already removed. Beware of this as you attempt to add assembly constraints to the part in the assembly.

Benefit of using Adaptive Sketches with Cross Part Sketch Geometry

- Mating geometry does not need to be recreated and/or updated separately when changes are made.

Concerns with using Adaptive Sketches with Cross Part Sketch Geometry

- If a change occurs while the adaptive sketch is temporarily set to non-adaptive, the change may not appear when the adaptivity is turned back on. The Rebuild All command found in the Tools menu must be run for the change to appear.
- If an entity in a multiple entity Reference object is removed the Reference link will be broken.
- If an entity is added to a face, used in a multiple entity Reference, the Reference will be updated in the sketch. However, the profile selection of any feature using that sketch will need to be updated manually.
- If the entity within a single entity Reference object drastically changes or is removed a "Cross Part association has failed" error (see below) could occur.



- To fix the error edit the adaptive part and right-click on the Reference icon below the sketch. Select Redefine from the context menu. By selecting new or changed geometry the reference object is updated with new projected geometry.
- Cross part sketches must occur between 2 parts at the same assembly level.
- If a part is Promoted or Demoted in the assembly hierarchy the link is permanently deleted.

- Parts with cross part sketches cannot be used to create iParts.
- Features with cross part sketches cannot be used to create iFeatures.

Adaptive Sketches – Underconstrained Sketch Geometry

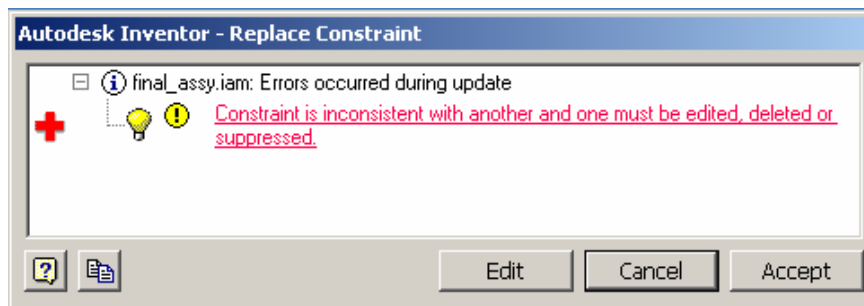
Another way to add an adaptive constraint to a sketch is to leave the sketch underconstrained. Once the part is added to the assembly and assembly constraints are applied the sketch will attempt to update and meet the assembly constraints. An example of this would be a linkage component. The length of the linkage depends on how it is used in an assembly. If the length is left unconstrained in the sketch, and the part and sketch are made adaptive, the part should update when constrained on both ends in an assembly.

To create an adaptive part using the Underconstrained Sketch Geometry option the user does not need to be in an assembly. The only difference between this and normal modeling practice is that part of the sketch will need to be left underconstrained. Again, it is best to plan ahead, having some idea of what needs to change in the assembly. In the example below, tangent and equal constraints along with 2 dimensional constraints have been applied. This fully constrains the sketch with the exception of the distance between the centers. Sketch constraints are left off so they will not conflict with the adaptivity constraints applied later. It is a good idea to drag the sketch to make sure it reacts predictably. The sketch can be set to adaptive at this point or later once it is consumed by a feature. Either way produces the same result.



The sketch is then used in a feature or shared and used in multiple features. To add an adaptive constraint the part will need to be placed in an assembly. The part is then made adaptive in the assembly by right-clicking the icon in the browser and selecting Adaptivity from the context menu. This makes the part adaptive, but Inventor needs to know which feature in that part is adaptive as well. If the feature and sketch were not previously set to adaptive it can be done now by editing the part and right clicking on the feature to make it adaptive.

Once the part, feature, and sketch are set to adaptive, assembly constraints can be applied. Inventor will attempt to make the assembly constraints work without adapting the part first. If the assembly constraints will not work, Inventor will attempt to adapt the part to make them work. At this point it is usually obvious if the sketch was not constrained to react predictably. If the sketch was not constrained correctly an assembly constraint error (see below) could occur. It is also possible that the sketch could adapt differently than you intended.



Benefits of using Adaptive Sketches with Underconstrained Sketch Geometry

- Parts can be designed based on their function inside an assembly without the user having to run the measure command, write down the dimension, open the part, and then add a constraint based on that dimension.
- Parts will update when either the size and shape or the placement of other parts in the design change.

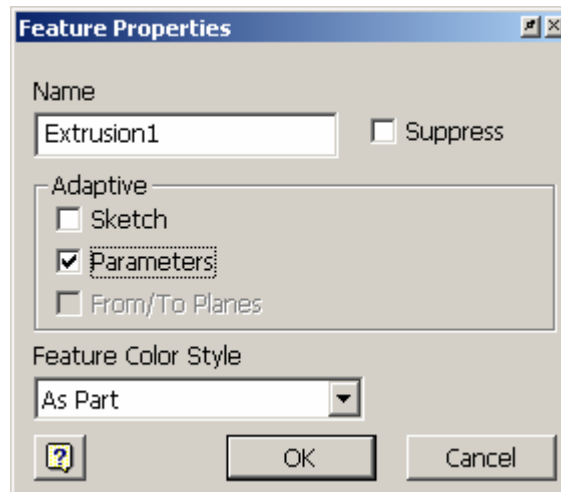
Concerns with using Adaptive Sketches with Underconstrained Sketch Geometry

- The sketch must be tested for predictability.
- Extra steps must be taken to make the part, feature, and sketch adaptive.

Adaptive Features – Adaptive Parameters

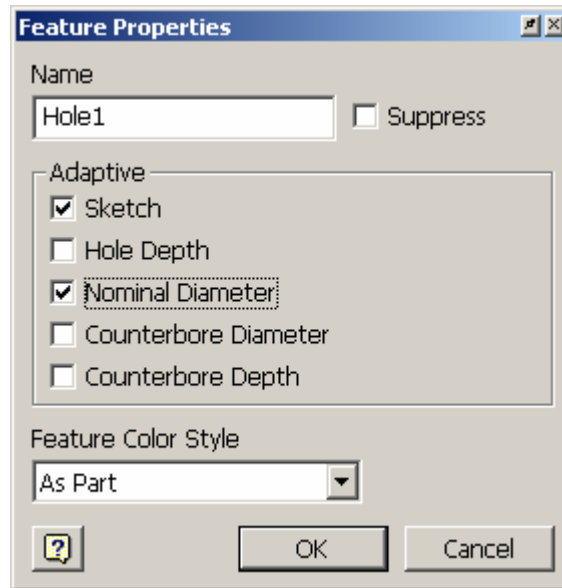
It is also possible to adaptively constrain parameters of a feature other than the sketch. This is helpful if the extents of an extrude feature needs to adapt inside an assembly or if the diameter of a hole needs to update when the bolt size changes.

To set up this type of adaptivity on an extrusion, simply right click the feature in the browser and select properties from the context menu. In the Feature Properties dialog (see below) select the Parameters option under Adaptive.



This tells Inventor that although the extent of the extrusion was defined when it was created it is now unconstrained and can have an adaptive constraint applied to it through an assembly.

When a feature is made adaptive from the context menu all properties in the Feature Properties dialog box are checked. The dialog box will differ depending on the type of feature selected. Below is the dialog box for a Hole feature. Notice it contains a lot more options for adaptive parameters.



Once the adaptive parameters properties are set, the workflow is much the same as adaptive sketches using underconstrained sketch geometry. Assembly constraints, which force the part to adapt, must be applied.

Adaptable Properties

Extrude

- Sketch
- Parameters
 - Distance Extents
- From/To Planes

Revolve

- Sketch
- Parameters
 - Angle of Revolution

Hole

- Sketch
- Hole Depth
 - Must be Flat Drill Point type with Distance Termination
- Nominal Diameter
- Counterbore Diameter
- Counterbore Depth

Benefits of using Adaptive Features with Adaptive Parameters

- Parts can be designed based on their function inside an assembly without the user having to run the measure command, write down the dimension, open the part, and then add a constraint based on that dimension.
- Parts will update when the size, shape, and/or placement of other parts in the design change.

Concern with using Adaptive Features with Adaptive Parameters

- Extra steps must be taken to make the part and feature parameter adaptive.

Adaptive Work Features – Created in the Context of an Assembly

Work features can be created using geometry from another part when editing a part within the context of an assembly. These work features will be set to adaptive automatically and an assembly constraint will be added. If adaptivity is removed they will be changed to grounded work features.

Benefits of using Adaptive Work Features Created in the Context of an Assembly

- Work Features are quickly defined based on their placement in an assembly.
- Multiple child features, of an adaptive work feature, will automatically update without having to make each one adaptive and then constraining it.

Concerns with using Adaptive Work Features Created in the Context of an Assembly

- The process is so automatic it is easy for users to create adaptive work features without intending to do so.
- If the adaptivity ever needs to be removed the grounded work feature must be redefined in order to be parametric. This can be a frustrating process if there are dependent features.

Adaptive Work Features – Created in a Part, Constrained in an Assembly

Work features created within a part, based on geometry in the part, can be made adaptive the same way other features are made adaptive. The difference is once a work feature is adaptive it becomes completely unconstrained regardless of how it was created. For example, a work plane may be created by selecting a face, dragging, and applying an offset value. If that work plane is made adaptive the complete definition of the feature is adaptive, not just the offset distance as one might expect. The adaptive work feature will be updated by adding assembly constraints.

Benefit of using Adaptive Work Features Not Created in the Context of an Assembly

- Multiple child features, of an adaptive work feature, will automatically update without having to make each one adaptive and then constraining it.

Concern with using Adaptive Work Features Not Created in the Context of an Assembly

- If the adaptivity ever needs to be removed the grounded work feature must be redefined in order to be parametric again. This can be a frustrating process if there are dependent features.

Adaptive Subassemblies – Subassembly Containing Adaptive Parts

It is possible in an assembly to specify a subassembly as being adaptive. One reason for this is to allow adaptive parts within that subassembly to adapt to any assembly constraints placed on them in the higher-level assembly.

Benefit of using Adaptive Subassemblies with Adaptive Parts

- The adaptivity of a part can be used regardless of what assembly level it resides in.

Concern with using Adaptive Subassemblies with Adaptive Parts

- Another file is required in the update of the adaptive constraint.

Adaptive Subassemblies – Underconstrained Subassembly

Normally when subassemblies are placed in a higher-level assembly they are treated as rigid bodies regardless of how they are or are not constrained in their own file. Identifying a subassembly as being adaptive allows underconstrained parts in that subassembly to reposition based on assembly constraints placed in the higher-level assembly.

Using adaptivity in this manner is no longer recommended. The release of Inventor 9 introduced the Flexible assembly option, which is a better-suited tool for this problem. However, it is still important to note that underconstrained parts in subassemblies, which are made adaptive for other reasons, will adjust based on assembly constraints.

Using Adaptivity with Drive Constraints

If adaptive constraints have been applied, using any of the methods that rely on assembly constraints, the Drive Constraint command can be used to animate that adaptivity. This is usually used for presentations showing springs, rubber components, or subassemblies reacting in an assembly.

Again, for the drive constraint to work correctly the adaptive sketch/feature/part/subassembly must rely on assembly constraints. Select the necessary constraint in the browser and right-click to select Drive Constraint from the context menu. In the Drive Constraint dialog box (see below) select the More (>>) button to show the rest of the dialog. Set all the necessary options to give the desired motion and check the Drive Adaptivity option. Notice that the Drive Adaptivity option and the Collision Detection option cannot be run at the same time.



Benefits of using Adaptivity with Drive Constraints

- Allows animation of parts changing dynamically based on movement in an assembly.
- Allows demonstration of springs or rubber components in use.
- Allows testing of adaptivity constraints for possible future changes.

Concern with using Adaptivity with Drive Constraints

- Uses a lot of system resources.

Common Concerns with Adaptivity

What happens to the adaptive part if the assembly is deleted or lost?

The adaptive part will remain in the last updated state. The “Adaptively used in assembly” setting will still be checked in the Document Settings. This prevents the part from being used adaptively anywhere else and being changed. If the part needs to be changed remove this setting. The part can then be set to adaptive in another assembly. Another way to change the part is to simply remove the adaptivity from the affected features.

Once a part and feature is set to adaptive in an assembly can I still change it?

Yes, if you open the part you can still change any sketch or parameter as if it were not adaptive at all. However, once the assembly is opened and updated the part will revert back to the adaptive constraints applied in that assembly.

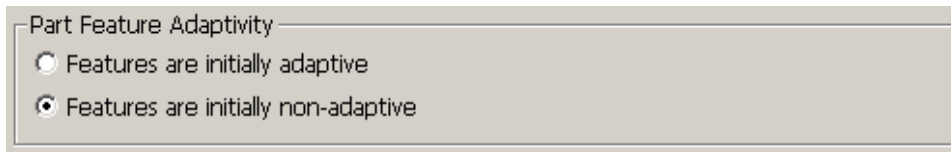
How does Adaptivity work with the Vault?

If an Adaptive part is in the Vault and is Checked In, Inventor will prompt the user to “Check Out” if a change is made in the assembly that changes the size and/or shape of the Adaptive part.

Tips and Tricks

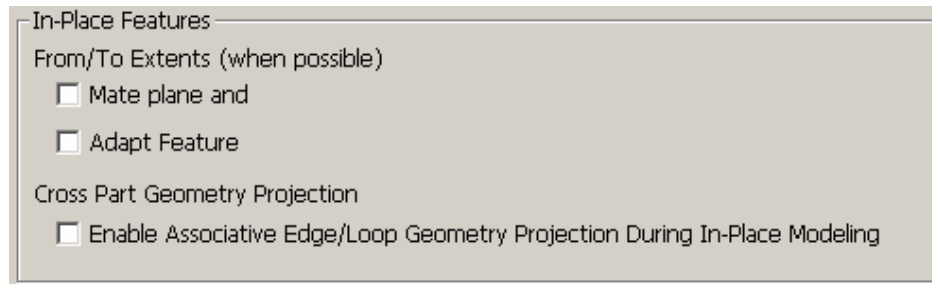
- Assembly constraint tips
 - Place the assembly constraint on the adaptive feature not on a child of the adaptive feature.
 - If a hole is adaptive, constrain to the hole not to a work axis created from the hole.
 - Use only one tangency per revolved feature.
 - Avoid offsets when applying constraints between two lines.
 - Avoid a mate constraint between two points, a point and a plane, a point and a line, and a line and a plane.
 - Avoid tangency between a sphere and a plane, a sphere and a cone, and two spheres.
- In assemblies with more than one occurrence of an adaptive part, constraints to nonadaptive occurrences may require two updates to solve correctly.
- Set Application Options (Assembly Tab)

- Part Feature Adaptivity



- Should be set to “Initially non-adaptive”
 - This forces the user to manually select which features can have adaptive constraints applied, giving them more control.

- In-Place Features



- From/To Extents (when possible) – Applies to using the From/To extents option in the extrude command and selecting faces or planes from other parts in the context of an assembly.
 - Mate Plane
 - When unchecked 2 non-adaptive work planes are created in the part at the location of the selected faces or planes.
 - When checked 2 non-adaptive work planes are created, same as unchecked, and Flush constraints are added in the assembly between those planes and the selected faces.

- Adapt Feature
 - When checked it reacts like the Mate Plane option except the new work planes are adaptive.
- Cross Part Geometry Projection
 - Enables automatic Cross Part Sketching when using the Project Geometry command while editing in the context of an assembly.
- Document Settings (Modeling Tab)



- Determines whether the current file has an adaptive occurrence in an assembly.
 - If checked no other occurrences of that file can be set to adaptive.
 - Problems can occur with this setting if a file is copied outside of Inventor. The setting will remain checked even though new copy is not used adaptively in any assembly. The user must open the file and clear the setting manually. If the file is copied from inside Inventor using the Save Copy As command the setting will be cleared automatically.
- Templates can be created with correct features already set to adaptive. Use the Create Component command and select the template. Once in the assembly, set the occurrence to be adaptive and add necessary constraints. A good example for this would be belts.

Contact Information:

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