Computer-Aided Products

KAP's Corner

Life-Long Mates

- **SolidWorks**
- COSMOS
- PDM
- Rapid Prototyping
- Office Productivity Tools

Mates are the "glue" that hold your assemblies together. One of the great strengths of Solidworks assemblies is that mates are independent objects which can be applied at any time, in any order. This makes assembly design easy for the CAD novice. It also means that a wide variety of mating strategies can be applied, based upon industry needs and user preference. As you become more expert at Solidworks, you discover that although Mates are relatively unconstrained, they are by no means unimportant.

The manner in which mates are applied have a strong impact on robustness – i.e., the survival of your mate scheme when changes are made. If you dread making changes to your parts or their mates, for fear that the Feature Tree will become festooned with those round, red error markers (I call that the 'Cherry Tree'), then this article is for you.

About KAP

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Keith Pedersen has a BSME from Clarkson College and an MSME from Boston University. After a stint at General Electric in Burlington, VT, Keith was the lead Applications Engineer for Advanced Surfacing products for Matra Datavision USA, including EUCLID-IS, UniSurf, and STRIM. He joined CAP in 1998 to support advanced surfacing applications in SDRC I-DEAS and joined our SolidWorks group one year later. Keith has extensive industry and consulting experience in non-linear Finite Element Analysis and Computational Fluid Dynamics in addition to surfacing applications. He is a Certified SolidWorks Professional (CSWP) and certified to train and support COSMOSWorks.

About CAP

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Introduction to Mate References

Most mates relate something to something else. The 'something' you select, be it a point, a line, or a face, is called a Mate Reference. The CAD system addresses these references by their index, which is largely the order that they occur in the datastructure. For example, a face reference might be identified internally as; "Face_3 of Extrude_7". It could be the 3rd face of this feature because it was generated by the 3rd line segment of the feature's base sketch.

If the base sketch of the Extrude_7 feature were to be edited (such that line segments were either added or removed) then the originally referenced face might change to, say, "Face_4 of Extrude_7". Any mate that was applied prior to this change would still contain a reference to "Face_3". However, that index now belongs to some other face! This is why major edits to a part file will sometimes damage your system of mates in any assembly using that part.

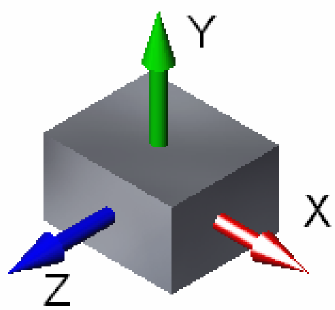
Adding or removing line segments to a base sketch are an example of a ***topological*** change. Another example would be reversing the direction of an extrusion, or changing the number of instances in a pattern. Topology changes usually result in re-ordering of model faces, edges, and points.

Parameter changes to the model may make features larger or smaller, but they usually do not change the number or order of model faces. This is why assembly Mates that reference model faces behave very well when edited parametrically. This happy fact is less true for mate references to model Edges. It is possible that a combination of parameter changes could cause faces to move or stretch so much that they intersect each other differently than originally planned, and so the number and/or order of model Edges has changed. So mate references to model edges are less robust than faces. By the same logic, the vertex points on a model are even less robust choices for mate references than edges. Points can come and go easily, especially as a result of drafting, filleting, or chamfering a part file.

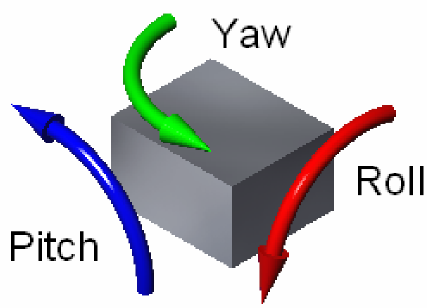
KAP's Tip: Throughout Solidworks in general, Face selections are smarter than Edges, and Edge selections are smarter than picking Points.

In the Solidworks Essentials class, we do not delve into the mechanics of the CAD datastructure. But we do advise students that it is generally best to create assembly Mates that reference faces. In summary, Faces are more robust than Edges, and Edges are more robust than Vertex Points.

Degrees of Freedom



3 Translations



3 Rotations

The simple rule about always mating to faces is a good starting point for novice users, for small assemblies, and for most simple, parametric revisions. As assemblies get larger, and involve more users and more evolutionary changes, a more sophisticated level of understanding of Mates becomes desirable.

Prior to Mate or Fix relations, every assembly component has 6 degrees-of-freedom (DOF). These 6 DOF are pictured at left. Mates remove one, two, or three DOF, and it usually takes 3 mates to fully constrain a part's location.

Let's apply 3 mates to the cube shown at left. First we mate the rightmost (+X) face to the assembly Right Plane, then the topmost face (+Y) to the assembly Top Plane, and finally mate the +Z face to the assembly Front Plane. This will certainly position the block rigidly in space. In fact, it will actually over-constrain the system. We have actually just restrained 9 degrees-of-freedom, not 6. Why?

Consider the very first mate to the +X face. It prevents the block from shifting (translating) in the X direction certainly, but it also eliminates two rotational degrees of freedom. After applying this first mate, the block can still 'roll' around the X axis, but cannot 'pitch' or 'yaw'. In fact, each of the face-to-plane mates restricts 3 DOF, one in translation, and two in rotation. Thus, this group of three face-to-plane mates will apply redundant constraints to all three rotation directions.

Does this matter? Not initially. As long as the walls of the block stay perfectly perpendicular, Solidworks will realize that the redundant constraints are 'consistent' with each other, not presenting a conflict. But as the assembly gets larger, the performance lost by carrying +50% extra restraints will start to add up. Also, consider what will happen to this system of mates when some of the model faces requires addition of a draft angle. Suddenly the overlapping restraints on the rotational DOF are no longer consistent with each other, and instead of flagging just one other mate as needing repair, the discontent will spread to every mate that seems to have a share in the affected degrees-of-freedom. This makes the isolation and diagnosis of problems more difficult.

Fortunately, there is a simple technique for applying Mates that avoids the creation of redundant constraints.

The 3-2-1 Method

The most efficient mating scheme you can follow is the same one used by machinists and inspectors to establish datum planes on a part. The Primary datum is usually a plane (often determined by 3 datum target points). The Secondary datum should be a line (or two target points), and the tertiary datum is determined by a single point. This system restrains first 3, then 2, and finally just 1 DOF, uniquely locating the part in space.

KAP's Tip:

1st Mate: Face-to-Face

2nd Mate: Face-to-Edge

3rd Mate: Face-to-Point

The equivalent technique within Solidworks is to make your first mate a Face-to-Face (coincident). The second mate should be Face-to-Line, and the third should be a Face-to-Point mate.

The selection of these references requires some careful thought. Choose a face, edge and point within the part file that are most representative of how the part aligns at assembly. Also, pick geometry that is fundamental to the design, features that are not likely to vanish or alter significantly under future edits, or when finishing details like draft, fillets, and chamfers are applied.

For small assemblies, and especially those involving static parts, the savings afforded by this method are small, primarily affording easier edits. When an assembly has a large number of moving parts, however, the 3-2-1 method can become a make-or-break technique. When you have long sequences of connected parts, especially when connected in non-orthogonal chains, you can fall prey to round-off error.

Round-off is a dirty word in the CAD industry – something that is always present, but we don't like to talk about it. You could have a system of parts and mates that, in theory, should be perfectly sound, and yet the system flags mate errors. In another installment of KAP's Corner, I'll treat solutions to this problem in more detail. But for now, just imagine that every redundant mate in your assembly stack-up is an opportunity for the system to compute two different paths to the same end result. And if these two different paths come up with slightly different answers, even to within 20 or 30 millionths of an inch, the Mate solver will flag the entire system as inconsistent.

KAP's Tip:

Eliminating mate redundancies eliminates sources for artificial errors.

The Trump Card - Reference Geometry

O.K., time to recap. In the first section, we explained that Faces are smarter than Edges, and Edges are smarter than Points. So, always mate to Faces.

Then in the previous section, we said you should mate using the 3-2-1 method. So, always mate to a Face and an Edge and a Point. But isn't that bad – aren't Edges and Points unreliable references?

The answer is - not all points and not all lines are unreliable. In fact, there are some lines and points in your model that are rock-solid. They are the points and lines lying inside the sketches that make up your model features. This is a powerful and often overlooked feature of Solidworks - that we can mate to reference geometry, as well as to the visible model boundary. Simply right-mouse-click over the icon for any feature's sketch, select *SHOW*, and then create your mates. Once finished mating, you can *HIDE* the sketch again.

KAP's Tip:

*Don't pick model edges or vertices as a Mate Reference, if you can instead select Points and Lines from **within** a feature's original sketch.*

The best mate reference of all is to select Planes. The Front, Top, and Right plane that start every Part file are immutable. User-defined reference planes can also be very good for mating against, as are reference axes.

Sometimes users will create sketches (either 2D or 3D) solely for the purpose of locating points and lines in space to serve as mate targets. They will generally name these sketches by some convention that helps other users understand their utility, so that they do not get edited or suppressed without first considering the impact upon parent assemblies.

So we must amend our description of the 3-2-1 method somewhat. Whenever I spoke of mating to a Face before, simply substitute the phrase "Plane or Face", with a preference for choosing a plane. Whenever I spoke previously of mating to an Edge, now substitute "Axis or Sketch Line or Model Edge" in that order of preference. Finally, whenever mating to a Point, 3D Reference Points or Sketch Points should be taken in preference to model Vertex Points.

Conclusion

If you have made extensive use of reference geometry for your mate references instead of picking model edges and vertex points, then your model becomes much more configurable. That is, you can create configurations that convey different levels of geometric detail or that represent different sizes in a family of parts – and the assembly mates should all hold up as you switch between different part configurations.

Also, if you need to make a part substitution and the original part has reference planes or sketches that are identified as mate references, it will be fairly painless to provide the newer version of the part with the same construction geometry, swap it into the assembly, and re-direct the mates – even if the new part has a completely different build history.

KAP's Tip:

Use the 3-2-1 method to create robust assemblies and ensure the survival of your mate scheme through design iterations.

Wherever possible, mate parts using the 3-2-1 method. This creates the fastest, most efficient constraint systems, and they will be less prone to stack-up of round-off errors. This also makes it far easier to diagnose and repair mates after making major part changes.

Within the 3-2-1 method, remember that:

- Planes are smarter than faces
- Axes and sketch lines are smarter than model edges
- Reference points and sketch points are smarter than model vertexes

The manner in which mates are applied have a strong impact on robustness – i.e., the survival of your mate scheme when changes are made.