



PTC Product Focus: A) [Pro/TOOLMAKER 5-Axis Option](#)
B) [New Product Introduction using Windchill ProjectLink](#)

Tips of the Month: A) [Comparing Parts, Assemblies and Drawings in Pro/ENGINEER](#)
B) [Creating and Reusing NPI Templates](#)

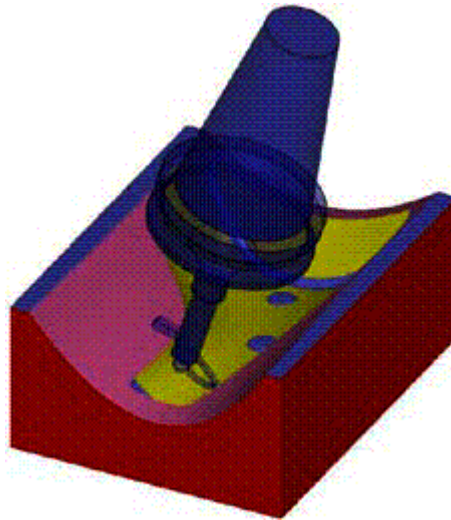
Announcements: [Most Recent Announcements](#)

Upcoming Events & Training Schedule: [Events & Training Schedule](#)

PTC Product Focus

Pro/TOOLMAKER 5-Axis Option

Easily leverage 5-Axis Power for Complex Geometries



Today's toolmakers are facing tremendous pressure to produce increasingly complex, higher-quality products, while reducing costs and production times. Pro/TOOLMAKER is a stand-alone solution that's ideal for shop floor programming.

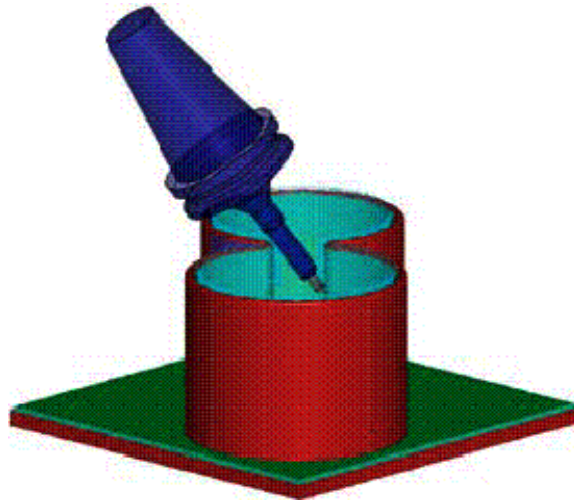
Pro/TOOLMAKER 5-Axis Option provides mold and die manufacturers with industry leading, high-speed precision machining for the most complex of geometries. Since it's fully integrated with PTC's Pro/ENGINEER software, you can be assured of its compatibility, reliability and ease-of-use.



Now you can take advantage of today's innovative, 5-axis CNC machines with Pro/TOOLMAKER 5-Axis Option—an advanced, yet easy-to-use application that lets you deliver higher-quality, highly complex parts faster. Pro/TOOLMAKER 5-Axis Option, a powerful add-on to Pro/TOOLMAKER, offers simultaneous 5-axis high-speed machining to help you optimize your tooling design and Manufacturing processes.

Key Benefits:

- Leverage high-speed machining—for even the most complex geometries
- Deliver products to market faster by reducing the number of machining setups
- Use shorter, more rigid cutters for higher feed-rates and optimized machining time
- Improve product quality through enhanced surface finishing capabilities
- Automatically converting 3-axis toolpath programs into 5-axis toolpath programs
- Pro/TOOLMAKER down milling, converted to 5-axis, to minimize 'C-axis' motion
- Further reduces costs and increases profitability by extending machine tool life

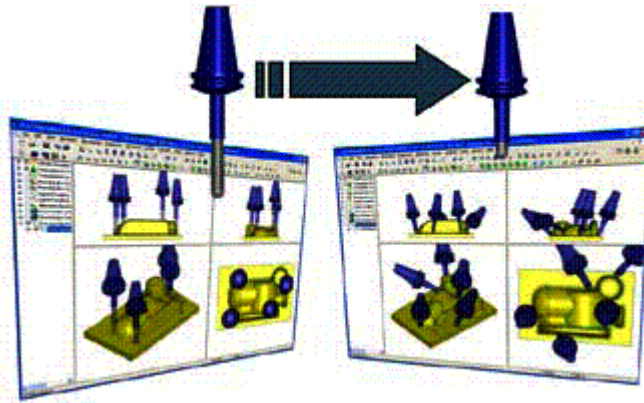


Key Capabilities:

Simultaneous 5-Axis Milling

Simplify designs using these highly automated toolpath creation processes:

- Automatic collision management
- Complete control over entry/exit, cut-to-cut, and between cut moves
- Full set of specialized surface machining strategies
- Swarf milling
- Electrode
- Morph
- Deep cavity/core machining
- Toolpath containment
- 3, 4 or 5-axis machining
- Thru curve tool axis option
- Attraction vectors
- Normal to surface
- Tilt/lead axis control
- Undercut milling with lollipop cutters



Post-processing

- Complete 5-axis post-processing solution with GPOST
- GPOST post-processor provides graphic NC post-processor generator, as well as interactive, online, context-sensitive help, and an extensive library of machine tools and CNC controls
- Standard APT output for existing post-processor system so you can use your Pro/NC posts

ProTOOLMAKER		Area Clearance Toolpath 1 [35x6, 0.5]		30/11/2008 10:02	
Cutter diameter	Corner radius				
35	6				
Tool name	Not Available				
Tool number	Not Available				
Tool type	Turntable				
Tool number	1				
Strategy	Area Clearance				
Ball length	5				
Taper angle	0				
Thickness	0.5				
Clamp	0.147				
Z clearance	73.875				
Wing over	0				
Wing over	2				
Spindle speed	2000				
Feed rate	100				
Rapid	10000				
Ramp down	7500				
Cooling	1000				
Ramp up	1000				
Min	Max			Tool file: C:\Program Files\PTC\Pro/TOOLMAKER\Toolpaths\Area Clearance 1 Project name: C:\Program Files\PTC\Pro/TOOLMAKER\Toolpaths\Area Clearance 1 Project name: C:\Program Files\PTC\Pro/TOOLMAKER\Toolpaths\Area Clearance 1 Tool file: C:\Program Files\PTC\Pro/TOOLMAKER\Toolpaths\Area Clearance 1	
X	1.40	1.07			
Y	4.0	1.03			
Z	1.717	73.875			

Data Interoperability

- Supports IGES, VDA-fs, STL, RAW, CLD
- SolidWorks® and Parasolid® (optionally available)
- CATIA® v4 and CATIA v5 (optionally available)
- Pro/ENGINEER® with GRANITE® kernel integrated in Pro/TOOLMAKER
- Model associativity for Pro/ENGINEER and SolidWorks models

[Back To Top](#)

PTC Product Focus

New Product Introduction using Windchill ProjectLink

[Click Here To View](#)

[Back To Top](#)

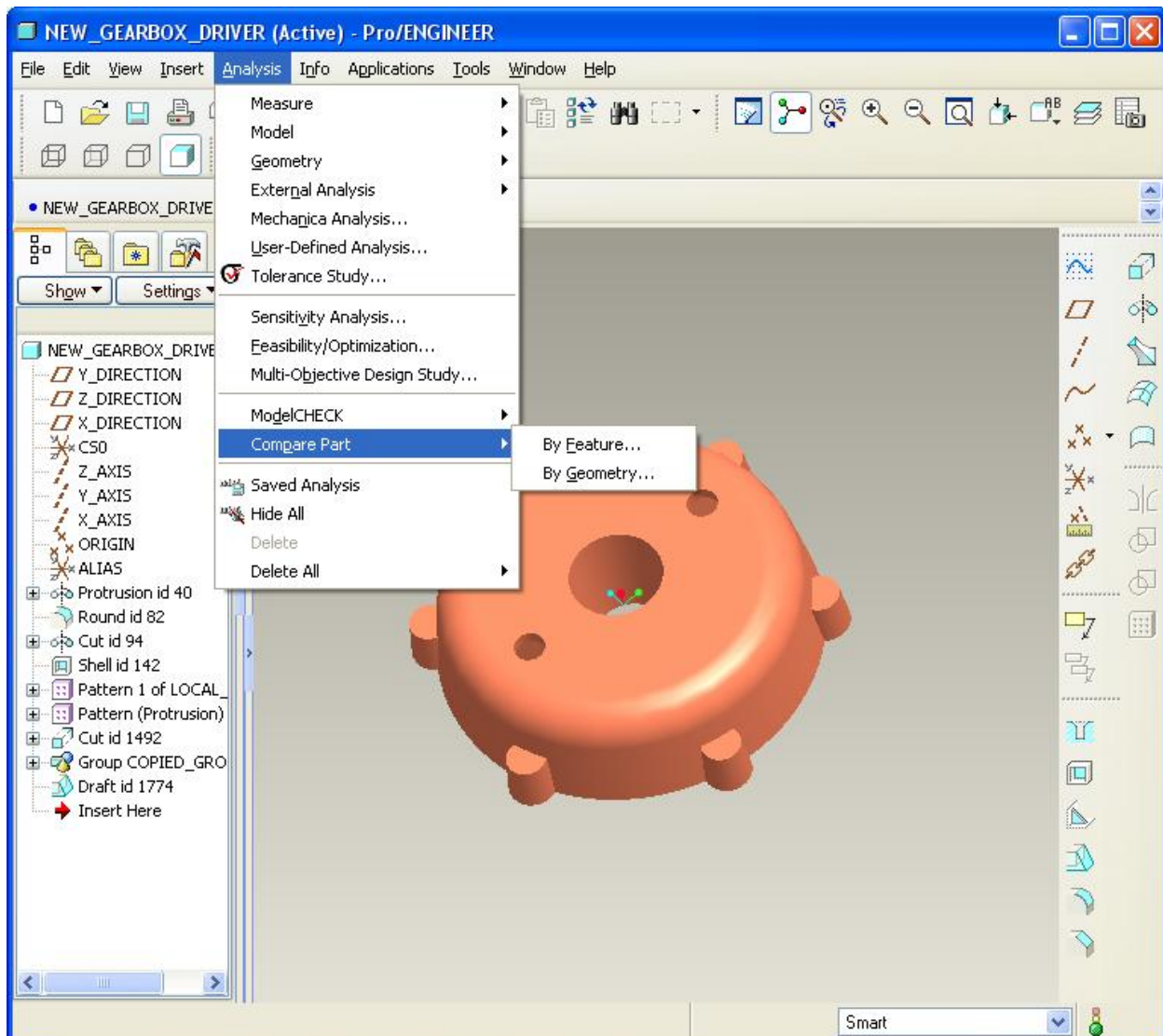
Tips of the Month

Comparing Parts, Assemblies and Drawings in Pro/ENGINEER

Did you ever receive two Pro/E models that appear to have the same geometry but they are really different? You run a mass property check on each model and the two are off by just a little bit but you can't tell where. If you are using PDMLink, you can use the Revision History to graphically show the differences but it is not an exact science. Visually comparing the differences is sometimes nearly impossible.

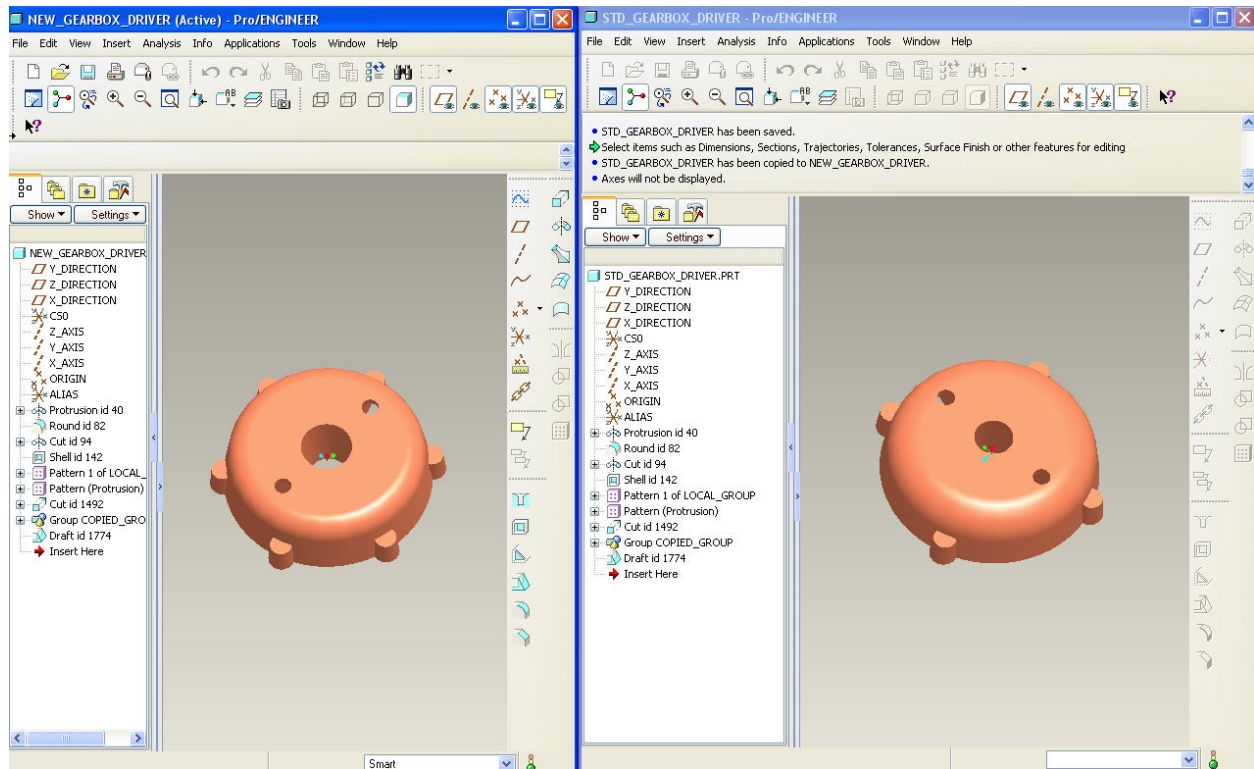
For quite some time now, Pro/E has had the ability to perform a Part Compare, Assembly Compare and Drawing Compare even if the objects are truly different models with the same geometry.

To access this functionality in Part Mode, you simply choose: Analysis / Compare Part.



You have the option to compare two models' features or geometry. Both methods come in handy depending on what situation you are in.

Let's say that we are working with a supplier who has had one of your models for quite some time. He gives it back to you and you want to know everything that he did to it while he had it. We decide to perform a Compare Part using the method outlined above.

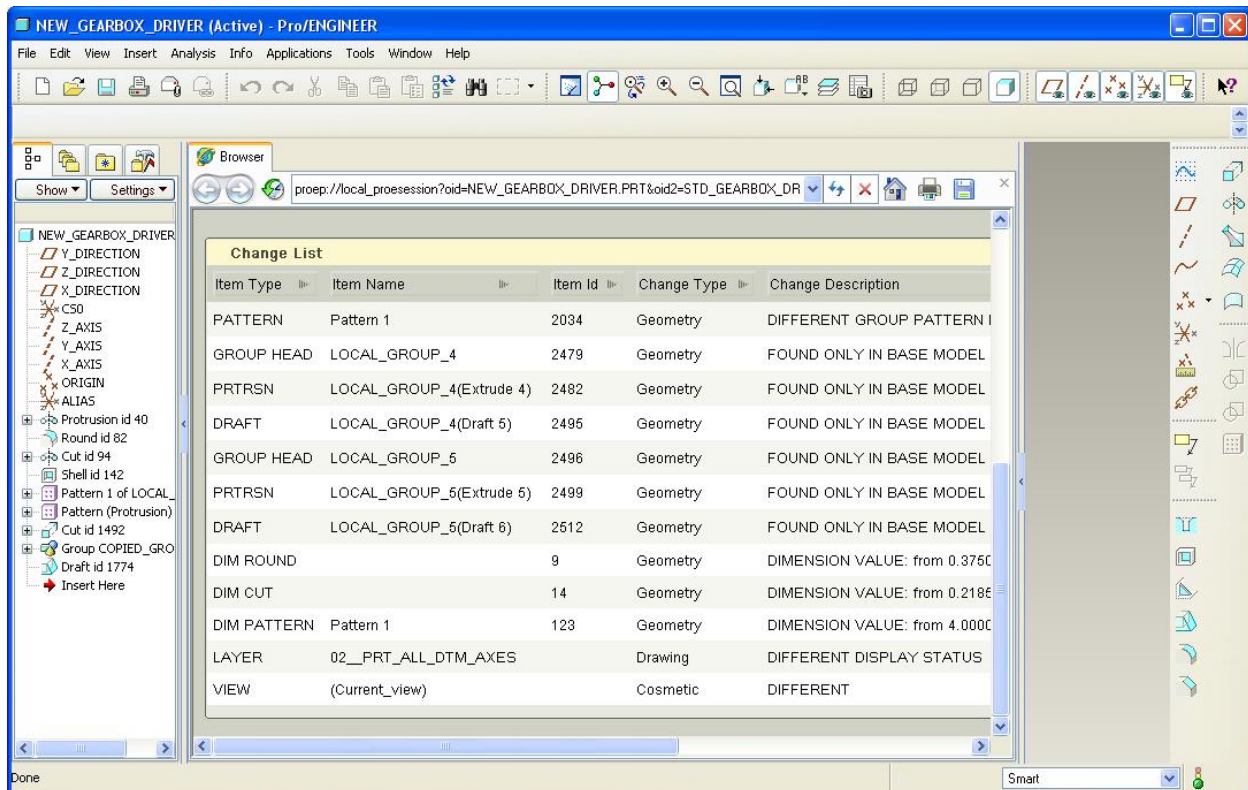


In the comparison, the original (or source) model may be on the left. The final (or target) model may be on the right. We can see that the model on the right has 4 protrusions around the outside instead of 6 but can you also see that the diameter of the hole in the middle has been modified?

If both models are Pro/E models a part comparison could tell you what features have been added, which ones have been removed and which have modified. The comparison also tells you information that you would not expect.

Any change made to the Pro/E model is tracked 100% because PDMLink or Pro/Intralink need to be able to create a new iteration for any and all model changes.

See the following which represents a comparison between the parts above:



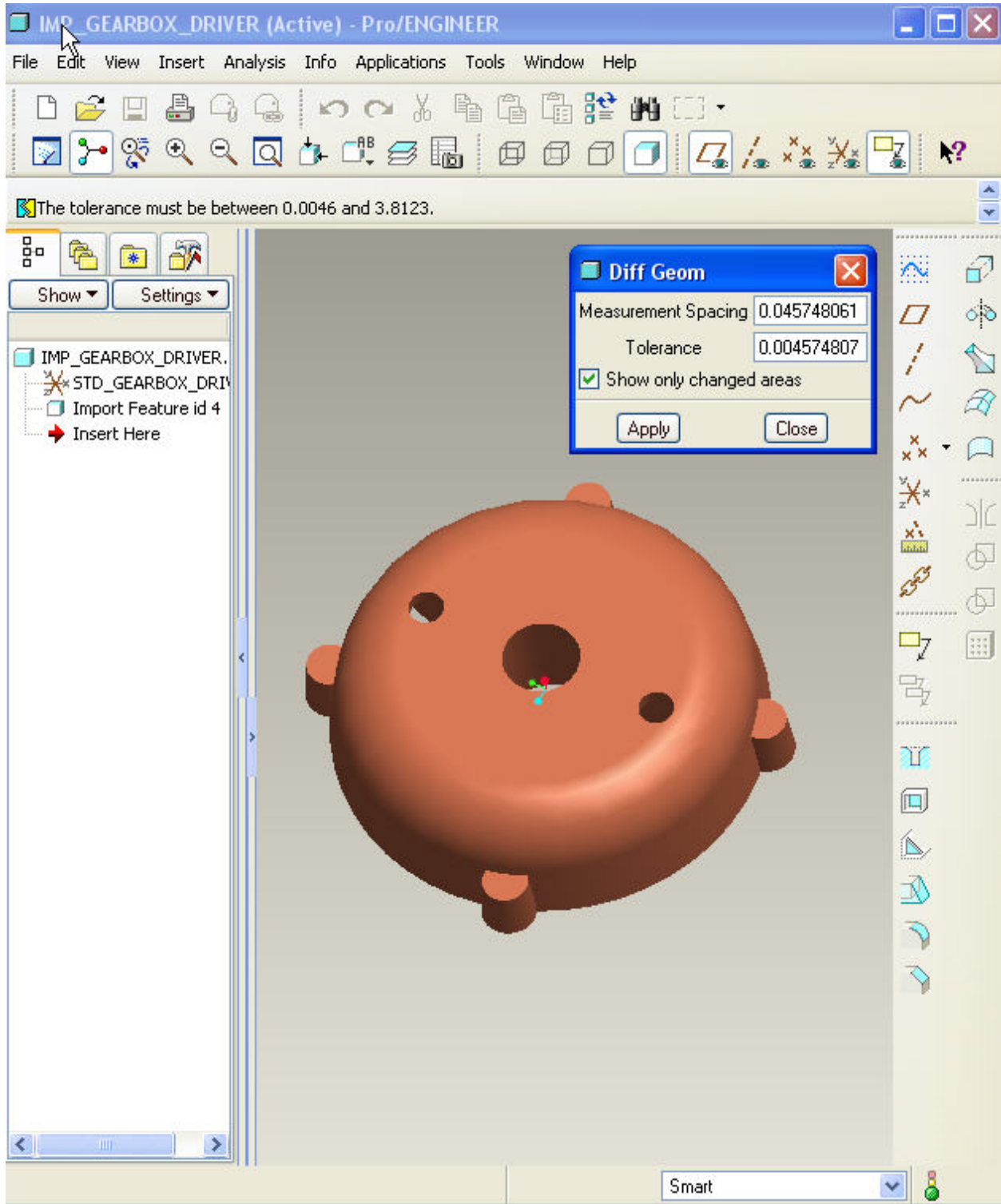
Notice that the comparison is quite granular. It shows added features, dimension modifications, and cosmetic items like the re-orientation of the model or the modification of a layer display.

Now we have a different scenario. We are in the process of migrating legacy CAD data into Pro/E. We could have had a third party company migrate the data for us so as not to interfere with our current workload or we could be rebuilding models ourselves that will require much future modification. The latter method is a good learning tool for customers that have recently moved to Pro/E. There are other methods to obtain a fully-featured or partially-featured migrated model but, for now, let's just use the simple re-authoring method.

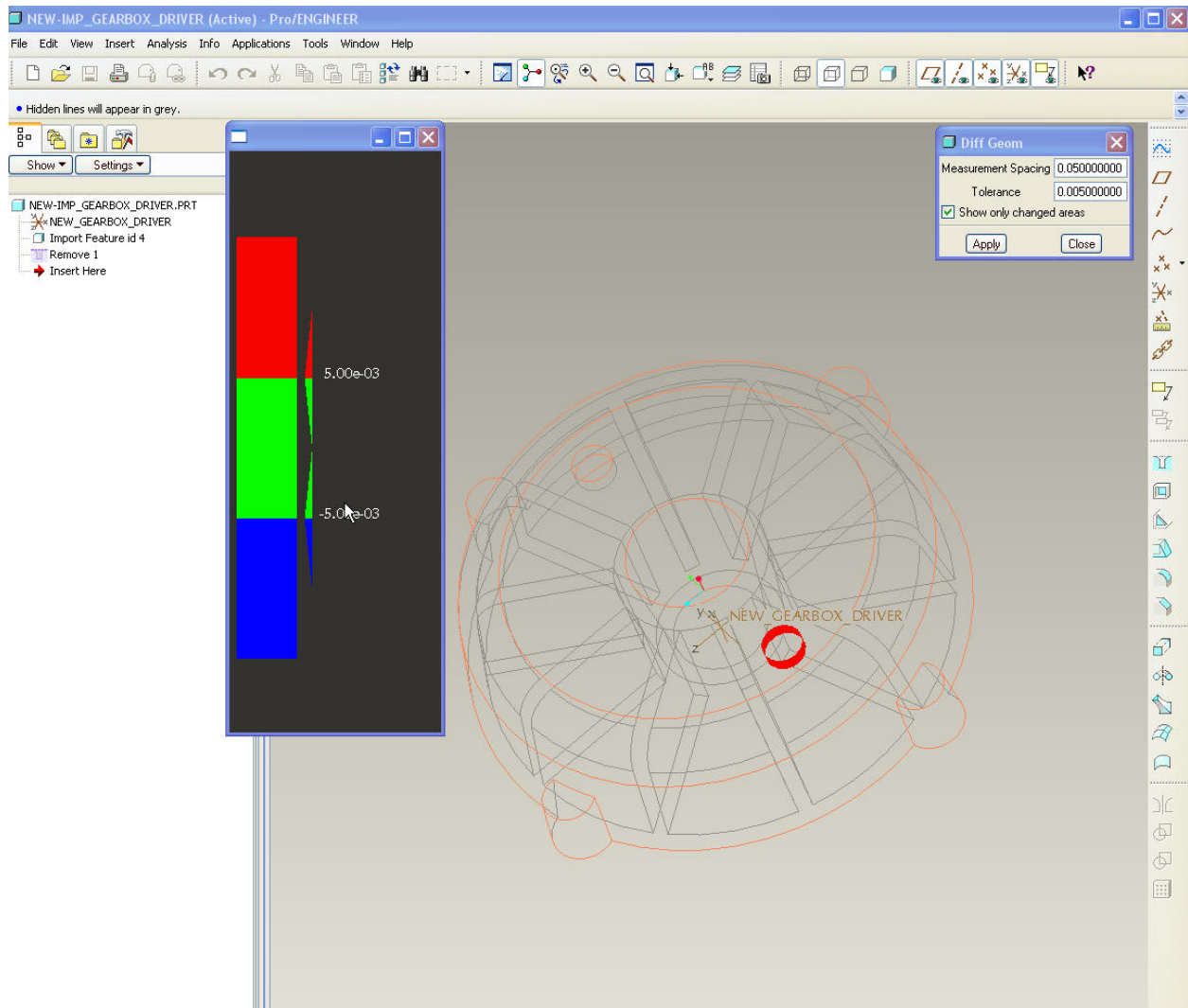
We have an imported version of the model coming from the legacy CAD system to compare our new model to. It has no features (because it was imported from a STEP file) so you will not be able to perform a feature by feature comparison as in the previous example. Here we will need to use the "By Geometry" option in the Compare Part functionality. While this method is not as granular as the previous method, it will tell you what surfaces are different on the feature-based model.

The next menu that comes up on the screen asks for values of "Measurement Spacing" and "Tolerance." There is actually a point cloud that gets applied to the surfaces of both parts. The measurement spacing refers the distance between points in the cloud. The Tolerance refers to the size of deviation that will indicate a difference. Pro/E returns two default values for measurement spacing and tolerance based upon model size. The best way to find the correct values for inputs is to simply type in a "0" for each. Obviously, this value is not valid so Pro/E will show you the range that you will be able to use. You want to have your inputs be as close to

the low end of these values as possible. The lower your inputs though, the longer your analysis takes. There will always be a tradeoff between analysis accuracy and time to compute.



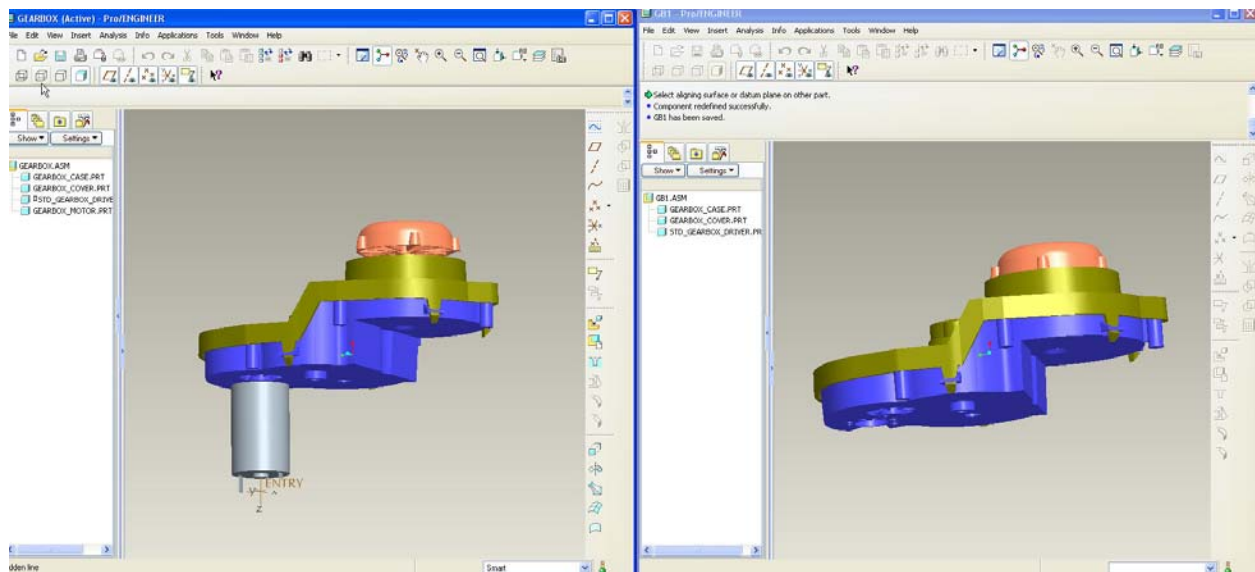
After choosing "Apply," Pro/E will calculate the differences between the models and return a graphical display of the geometric differences between the two models.



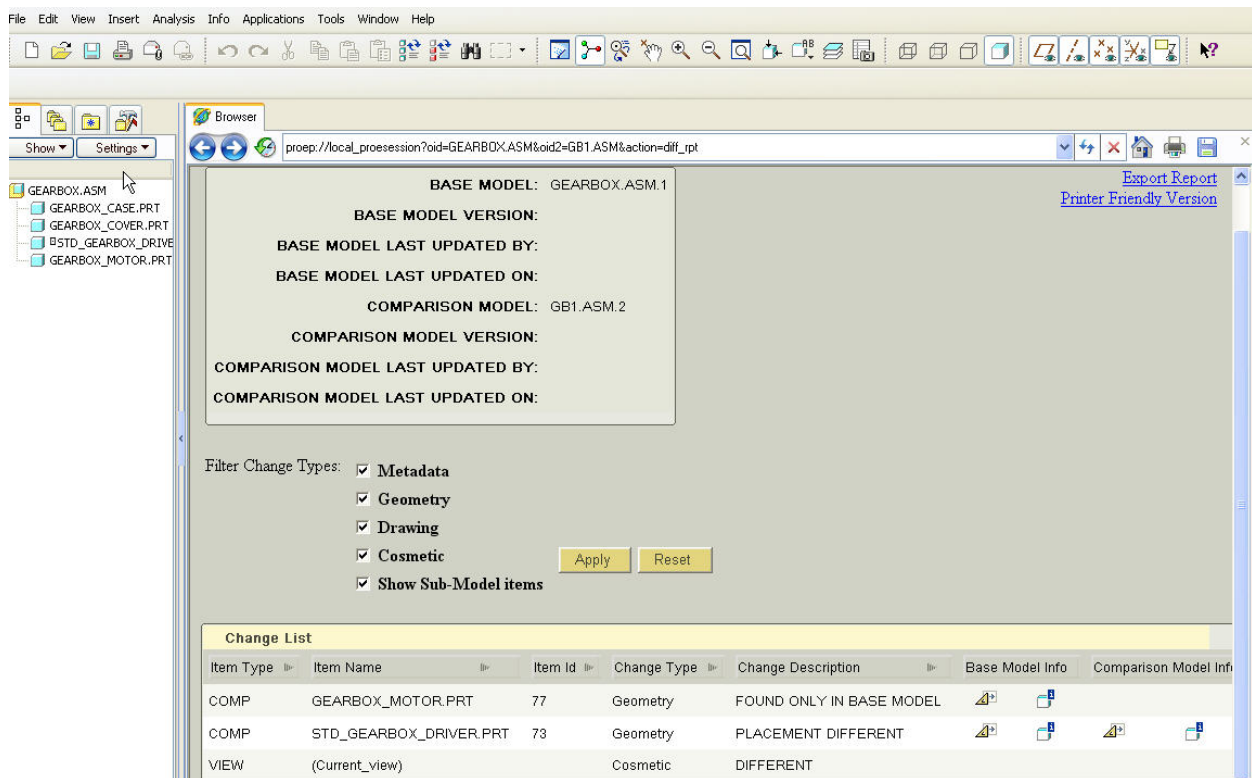
In the example above, the only difference between the two models was the presence of a hole.

The Compare Part shows this difference with a red surface.

The same technique can be performed for assemblies. In the following example, the same parts are used in both assemblies with a couple of exceptions. The cylindrical component is missing from the second assembly and the placement of the gearbox driver is different. Just like in the previous examples, leaving it to a visual comparison may not be adequate.

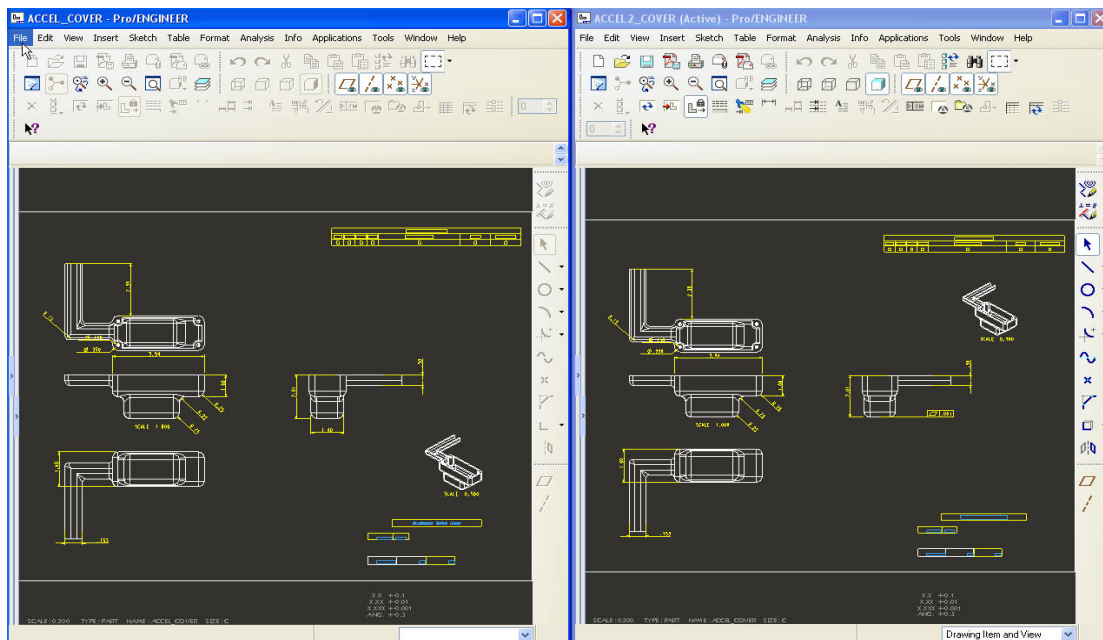


In an assembly, you can choose Analysis / Assembly Compare. You choose the assembly that you want to compare it to and Pro/E lists the differences. Below is a listing between these two assemblies:

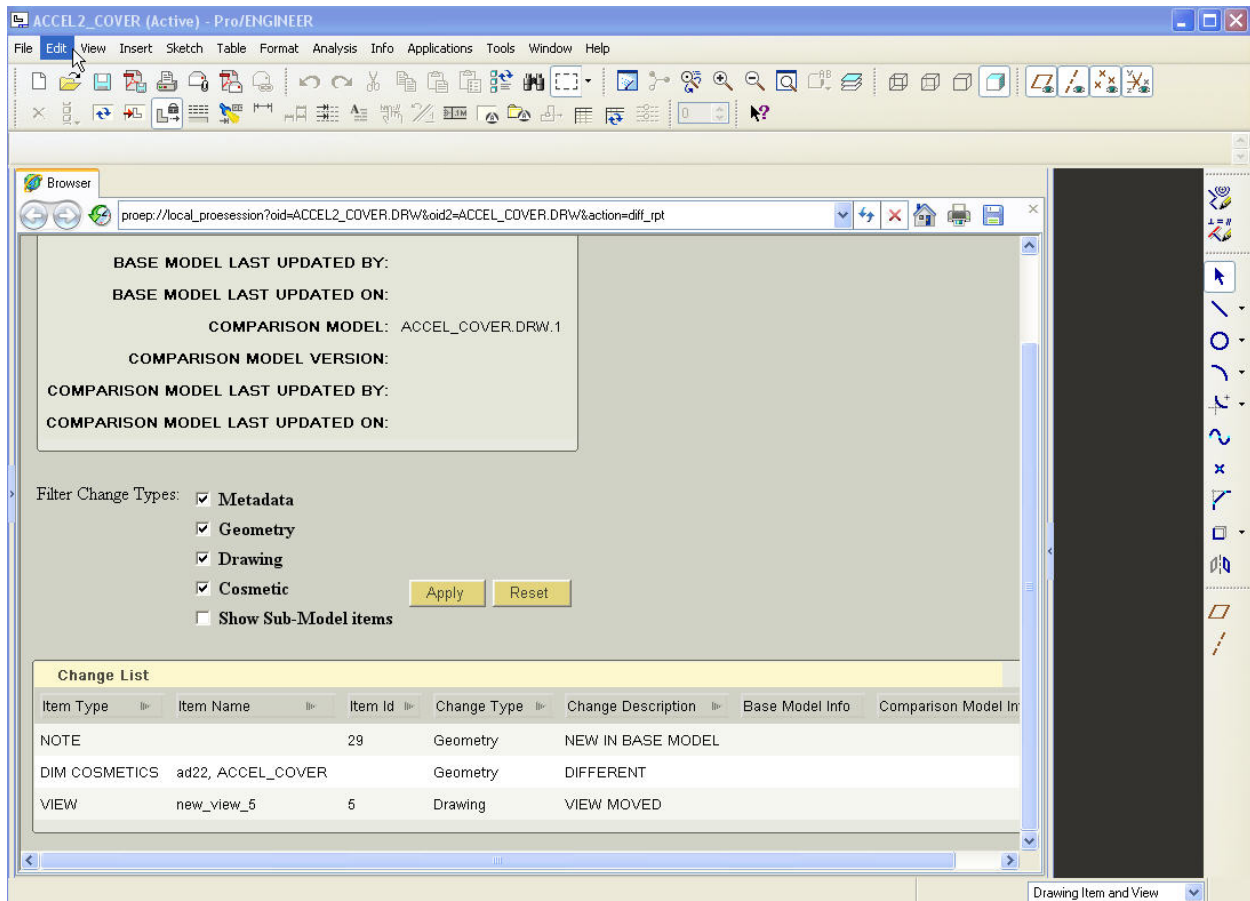


Here, you can see differences in placement, missing items, and even a modification to the current view. You can even choose the information buttons to list information or choose the small triangle to highlight these differences on the screen.

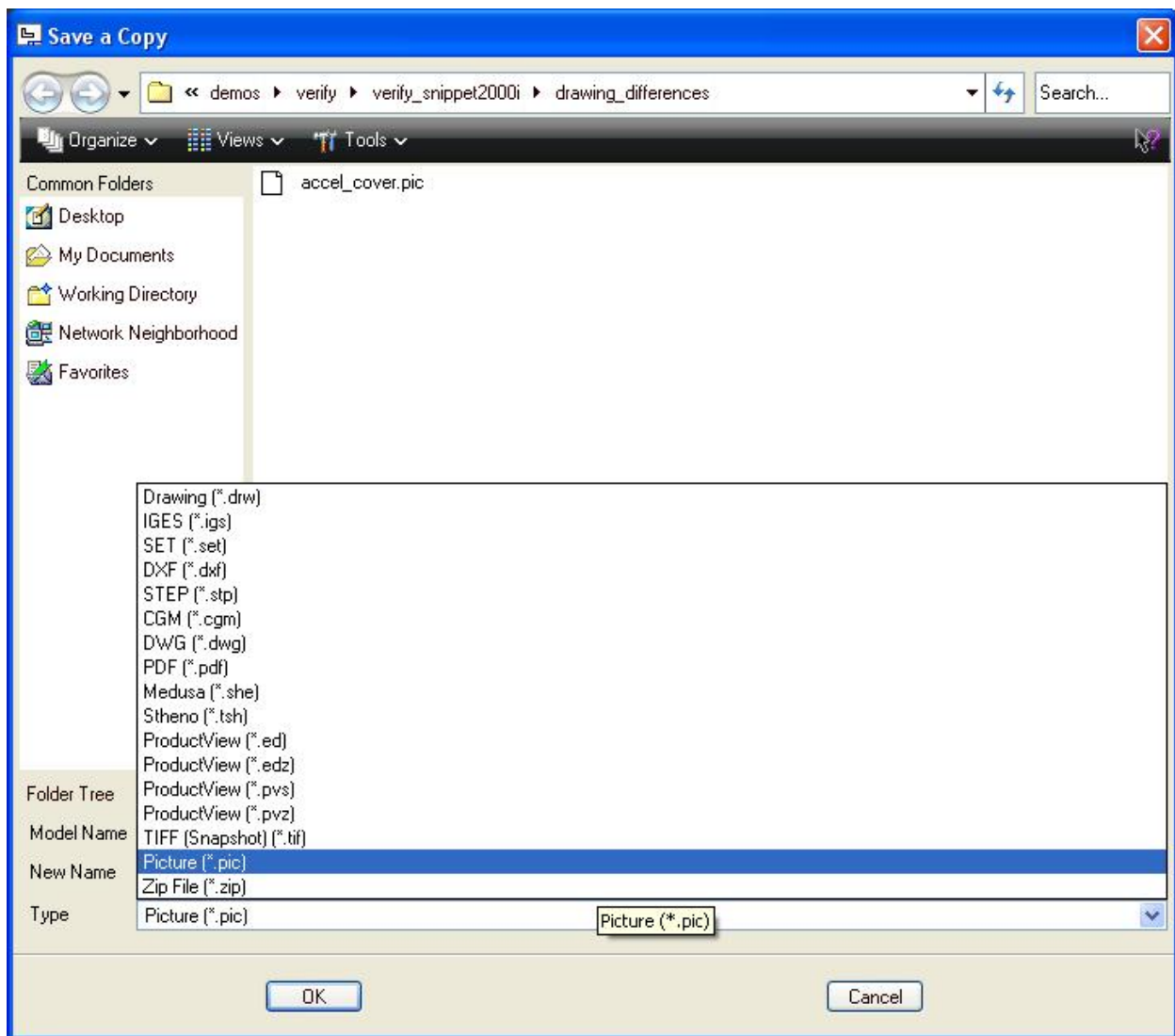
In the following example, we wish to compare the differences between two drawings.



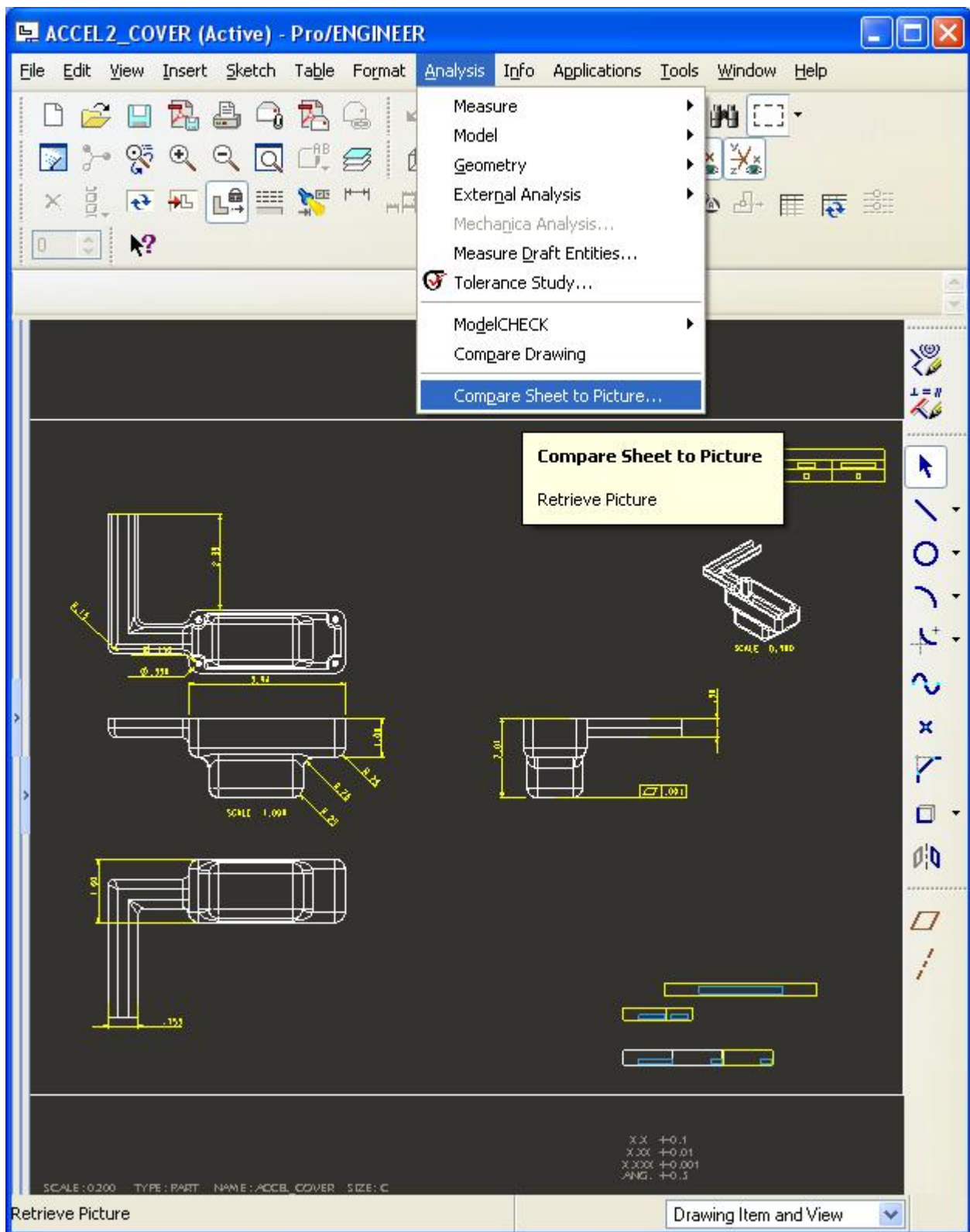
In many cases, the differences between two drawings can be even more difficult to determine than in parts and assemblies. Pro/E will list these differences in text form also.



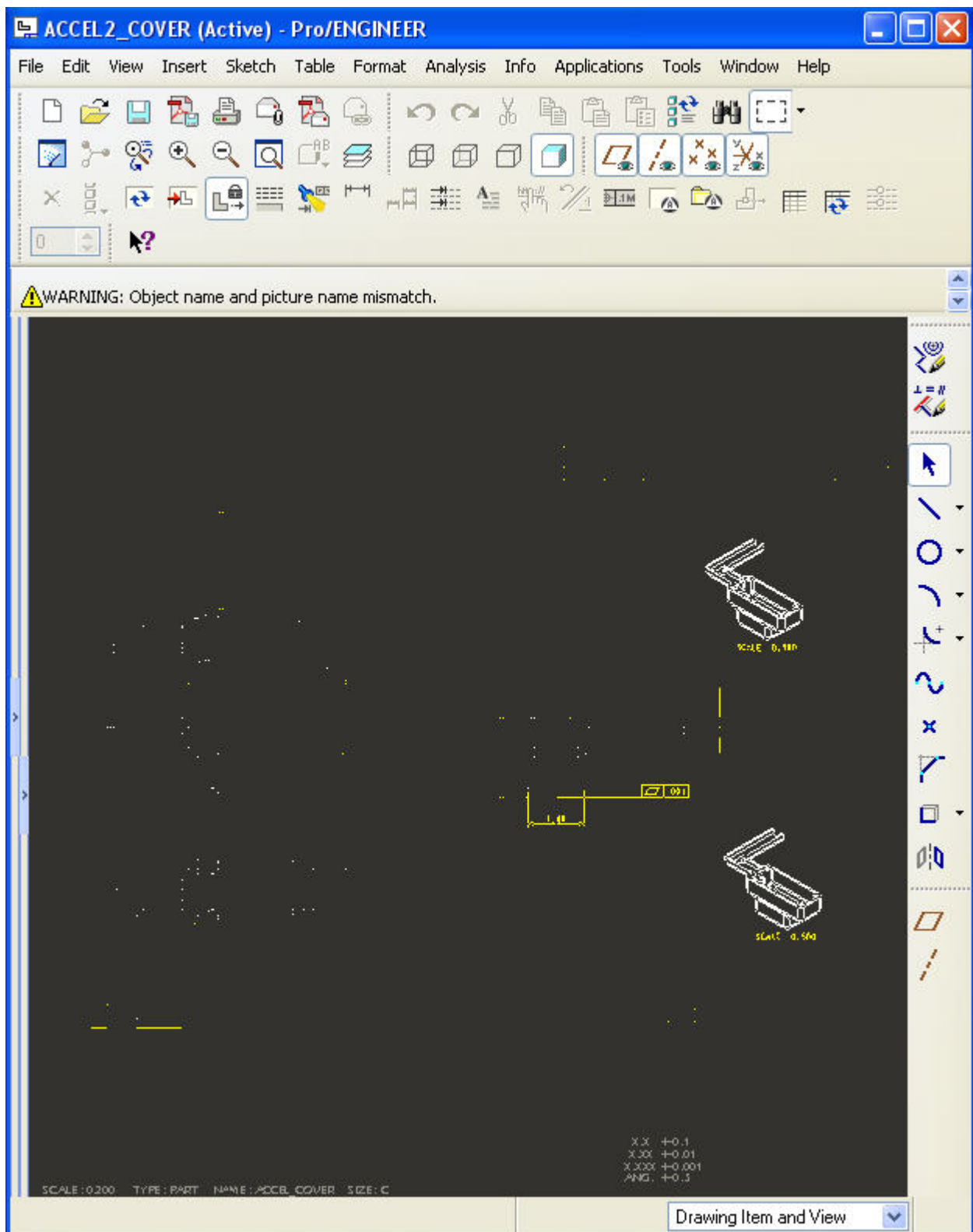
Note that there is a list of all of the differences. Unfortunately, there is nothing that graphically shows what these differences are. For this reason, there exists the ability to compare an existing drawing to a drawing picture file. By simply saving one drawing as a picture file (.pic), we can compare the second drawing to this image of the first.



Once you have this picture file, you can choose the other option for comparing a drawing:



The following shows the graphical differences between two drawings:



The above shows the movement of the isometric view along with the addition of a GT&T plus the removal of the shown dimension. There may also be some stray pixels but they can be ignored.

Note that additional methods exist to compare two drawings using ProductView Standard Edition. You can overlay two drawings showing one drawing in one color, the other drawing in another color, and the differences in a third color. In part mode, people also may have a need to compare a Pro/E model to an actual manufactured part. For this, you can use Pro/VERIFY to show deviations between a point cloud resulting from the laser digitizing of a manufactured part and a Pro/E model.

[Back To Top](#)

Tips of the Month

Creating and Reusing NPI Templates

[Click Here To View](#)

[Back To Top](#)

Announcements

Educational Resource Library

Learn things you always wanted to do - but didn't know you could.

This one stop educational resource library will help you learn more about PTC Solutions and provide you with technical materials developed by the product experts to help you become more productive.

Get tutorials, how-to videos and expert advice for:

- Pro/ENGINEER
 - Conceptual and Industrial Design
 - Detailed Design
 - Simulation/Analysis
 - Production
 - Design Collaboration
- Windchill PDMLink
- Windchill ProjectLink
- Pro/INTRALINK
- PTC Online Tools

Check out the [Educational Resource Library](#) today.

PTC Tips & Techniques Newsletter Archives

Miss an issue! Can't find that awesome technique you read about? Fear not, you can click on the link below and go through our Customer PTC E-Newsletter archives.

[Click Here To Access](#)

It's better than finding the Ark of the Covenant!

PTC Tips & Techniques Webcasts: Work Smarter. Not Harder.

Click below to see regularly scheduled Tips & Techniques technical Webcasts that are designed to provide you with the most popular time-saving tricks that Pro/ENGINEER users of all skill levels will find useful. Get more out of your maintenance dollars!

[Tips & Techniques: Work Smarter Not Harder!](#)

E-PROFILES IS HERE!!

We have been eagerly anticipating the debut of the new electronic version of Profiles Magazine and now it is here! This new web site will supplement the print edition of the magazine and will

provide new useful features not feasible with paper media. e-Profiles will provide you with 24x7, worldwide access to key information previously available exclusively in the print version. "Tips & Tricks," a popular feature pioneered by Pro/USER, has also moved to the web and will be expanded as the site matures.

Please take a few minutes to check out this new web site. We don't think you will be disappointed.

<http://profilesmagazine.com/>

[Back To Top](#)

Upcoming Events & Training Class Schedules

Upcoming, 2008/2009 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 7 – 10, 2009 Orlando, FL USA
PTC/USER World Event
<http://www.ptcuser.org/>

Events

Our seminars and conferences seek to provide you with relevant information regarding product development trends in your industry as well as innovative software learning experiences. Think of them as a constructive day off where you can share experiences and swap ideas with your peers.

If you can't manage to get away, we'll bring it to you. Check back often for regularly scheduled live webcast events.

[You're Invited to Attend...](#)

Please visit the [PTC Education Services](#) website for the latest training information including course descriptions, schedules, locations, and pricing.

- Attend a course at any PTC Center and receive a **free** copy of Pro/ENGINEER Wildfire Student Edition!

<http://www.ptc.com/services/edserv/index.htm>

Live Instructor-Lead Virtual PTC Training Courses

Virtual Classrooms provide interactive learning with a trained PTC instructor in convenient and manageable sessions that last approximately 4 hours over a series of days. It's easy to join a class right from your desk using a phone or voice-over IP technology.

Sessions are performed just like a traditional ILT (including interactive exercises where you and the instructor can work on lab exercises together) and feature some of our most popular ILT courses. These sessions cover the exact same material as the traditional ILT in-center courses. Also look for some of our most frequently requested mini-topics delivered in the same format that are only an hour - two hours in duration.

If you have any questions about these sessions or would like to see getting other courses, not on this list, on the schedule please feel free to contact me for more details. They are a great way to bring training to you without you having to worry about location or being out from work for long stretches.

You can register for these sessions just as you would for any normal ILT class either by:

1. calling order admin at <http://www.ptc.com/services/edserv/training/registra.htm> or
2. you can go to PTC University directly at <http://www.ptc.com/learning> and submit a registration request directly. All you have to do is search the catalog by typing in “virtual” in the search field and you will see a listing.

PTC

Note: This PTC E-Newsletter will continue to be used for the following:

- 1) Inform you on events related to PTC products (user groups, conferences, training schedules, etc.)
- 2) Educate you on solutions that are available at PTC
- 3) Tips & Techniques using PTC Products

Note: These messages are compiled in the local PTC office and will be distributed via e-mail.

[Back To Top](#)