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PTC Product Focus

Manikin Available with Pro/ENGINEER Wildfire 4.0

With the current release Pro/ENGINEER Wildfire 4.0 you can now add the ability to perform ergonomics and human factors analyses easily and quickly from directly within the Pro/ENGINEER assembly environment.

The name of the product is Pro/ENGINEER Manikin and provides an easy-to use human modeling solution that saves time and cost by reducing expensive physical prototypes, and makes the process of performing ergonomics and human factors analyses easily accessible to all [Pro/ENGINEER](#) users.

In the past, designers were forced to perform these types of studies by first creating a Pro/ENGINEER model and then exporting the model into a format that could be read by a third party human factors package where the study would be performed and the results would then have to be conveyed to design engineer to make changes in his Pro/ENGINEER design. This could often be a very iterative and time consuming process that was slow, potentially prone to error, required a high level of expertise, and often occurred too late in the design process.

By adding this capability directly into the Pro/ENGINEER design environment you can get early insight into how human interactions can be verified in your design. The results are immediate and can easily be performed as the design model progresses.

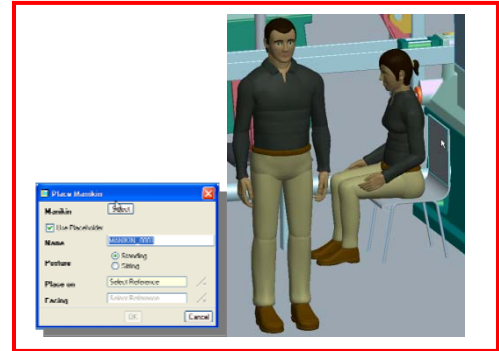
There are 3 levels of functionality that are available:

1. **Pro/ENGINEER Manikin Lite**
2. **Pro/ENGINEER Manikin Extension**
3. **Pro/ENGINEER Manikin Analysis Extension**

The **Pro/ENGINEER Manikin Lite** version will be available at no cost with every license of Pro/ENGINEER (Build M060 and above) and provide users with a limited set of functionality that is intended to allow them a chance to experience the ease of use, workflow, and benefits of working in the Manikin environment. Users will be able to select a single PTC Manikin, apply a limited set of postures, and perform manipulations to the Manikin.

Pro/ENGINEER Manikin Extension is the mainstream product that provides a desktop human modeling solution for first-level checks of human-centric form, fit and function. The capabilities include:

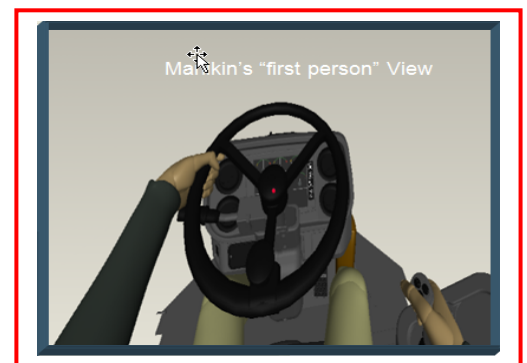
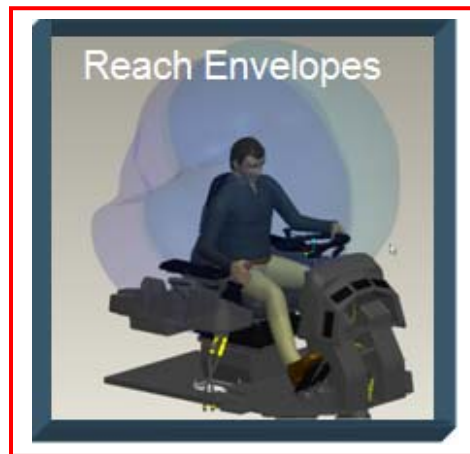
- Selection of Manikin from a predefined library of 5, 50, 95 percentile male, female, and child Global populations (Asia, Europe, Americas).
- Preset initial postures for standing or sitting.
- Placement options for orienting the Manikin.
- Fully modifiable library of Macro and Micro postures to choose from.
- Full Manikin manipulation



- Reach functionality
- Vision Cones



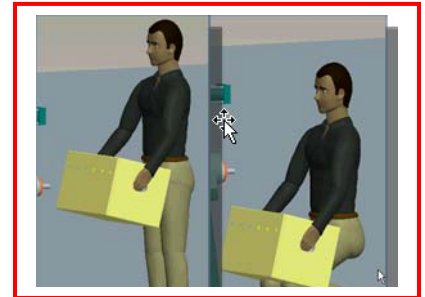
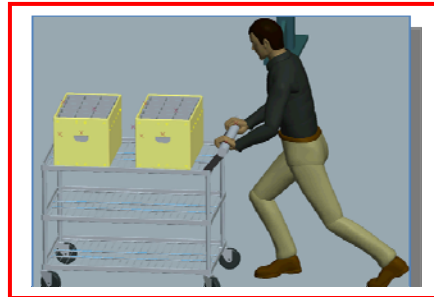
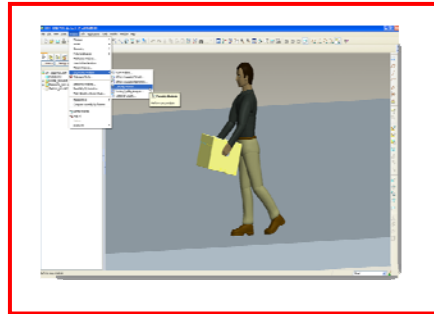
- Reach Envelopes
- "First Person" View
- Line of Site



Pro/ENGINEER Manikin Analysis Extension provides the ability to analyze human movements such as pushing, pulling, lifting, and lowering. The results are presented in the form of a report and stored in the model as an analysis feature in Pro/ENGINEER. By understanding the required forces for performing various tasks, you can optimize your design to help reduce operator fatigue and ensure safety in your products.

Some of the capabilities provided with the Manikin Analysis Extension include:

- Material Handling (RULA)
- Pushing/Pulling (Snook)
- Lifting/Lowering (Snook)
- Carrying (Snook)
- Lifting/Lowering (NIOSH)



If you would like more information on Manikin please visit our website at <http://www.ptc.com/products/proengineer/manikin-extension> or contact your PTC representative for a demonstration.

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PTC Product Focus

How Can IsoDraw Help Improve Your Aftermarket Sales?

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Tips of the Month

Options for Working with Imported Data

Pro/ENGINEER has many interface options that allow you to import many different file formats from a variety of sources. There are even direct translators available to many other CAD systems that allow Pro/ENGINEER to open them directly. But even with all of the options available, it doesn't guarantee that the file will import cleanly. Also, by default, the data that does come in cleanly (solid), will come in as one import feature in the model tree. So, what are your options for repairing your data that you import to get it into a solid, and what if you want to change that data to meet your design needs?

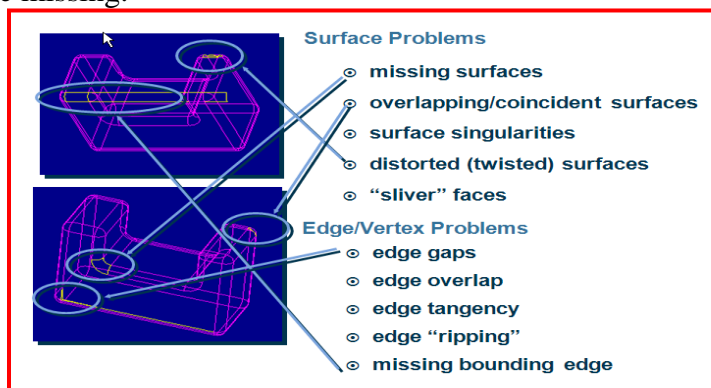
Wildfire 4.0 introduces several new tools and products that give you much more flexibility when working with imported data. Not only for repairing, but also for making changes to the data once it has been imported.

There are 3 main tools that can be used when working with imported solid geometry in Pro/ENGINEER. Depending on what the desired end use of the imported geometry is, will help you decide on which tool or combination of tools is right for you.

1) Import Data Doctor:

This is the application that provides the capability to repair 3D surface and solid data once it has been imported into Pro/ENGINEER. With the introduction of Wildfire 4.0, the entire Import Data Doctor repair tools have been rewritten to provide faster and more robust features for repairing data as well as added new functionality for modifying the data during the repair process.

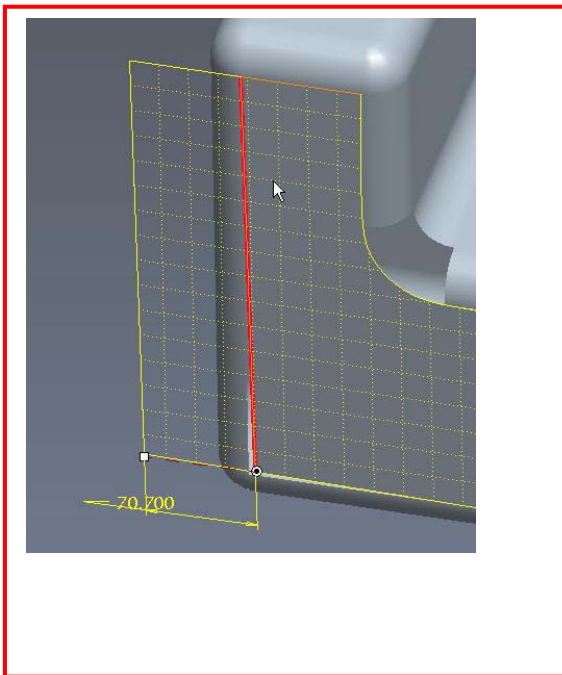
In the past, the only reason for entering the Import Data Doctor would have been to resolve a problem with the imported geometry. For example if a surface boundary was corrupted, surface was missing, extra surfaces present in the import, or any other problem that prevented the geometry from automatically coming into Pro/ENGINEER as a solid import. The Import Data Doctor still provides the necessary tools that will allow you to repair most any model that needs fixing using tools to zip up gaps, fix boundaries, repair edges, and even replace surfaces that are missing.



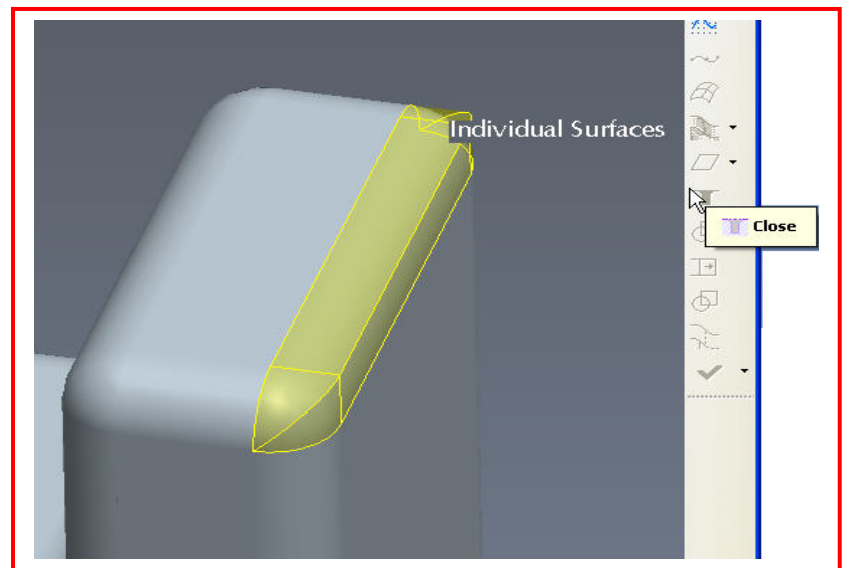
The new Import Data Doctor goes beyond just making the repairs. It also introduces the new functionality called “Featurize Mode” which gives you the ability to create “pseudo features” while in the repair environment.

Some of the capabilities introduced in Featurize Mode are:

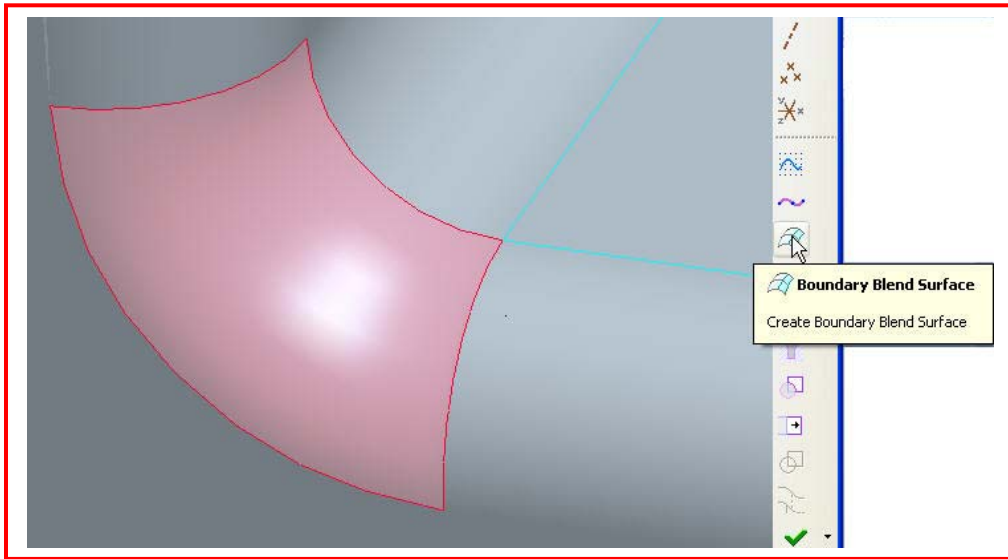
- Use the Boundary Blend feature while in the Import Data Doctor to add a missing surface. The resulting surface is automatically part of the import feature.
- Use the Surface Extend tool while in the Import Data Doctor to dynamically drag a boundary to extend it.
- Select surfaces and convert them to a Cylinder, Planar, Extrude, and Revolve features. This will allow you to reposition the geometry as a feature as well as **modify its dimensions** and even change its sketched section.
- Use standard selection techniques like Seed and Boundary to gather large groups of surfaces for conversion.
- Use the new Close tool to remove surfaces of geometry that are no longer needed and automatically close up the adjacent boundaries.



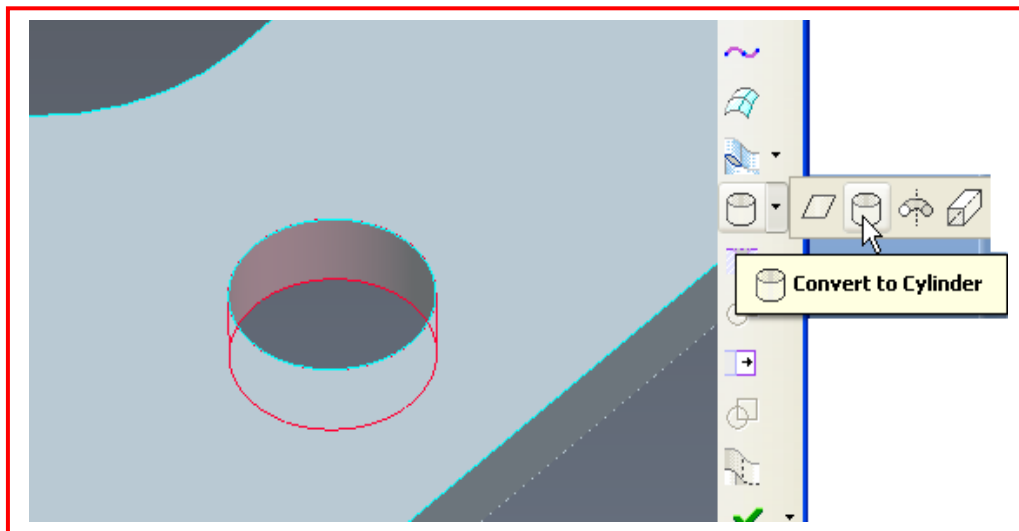
EXTEND



CLOSE



BOUNDARY BLEND



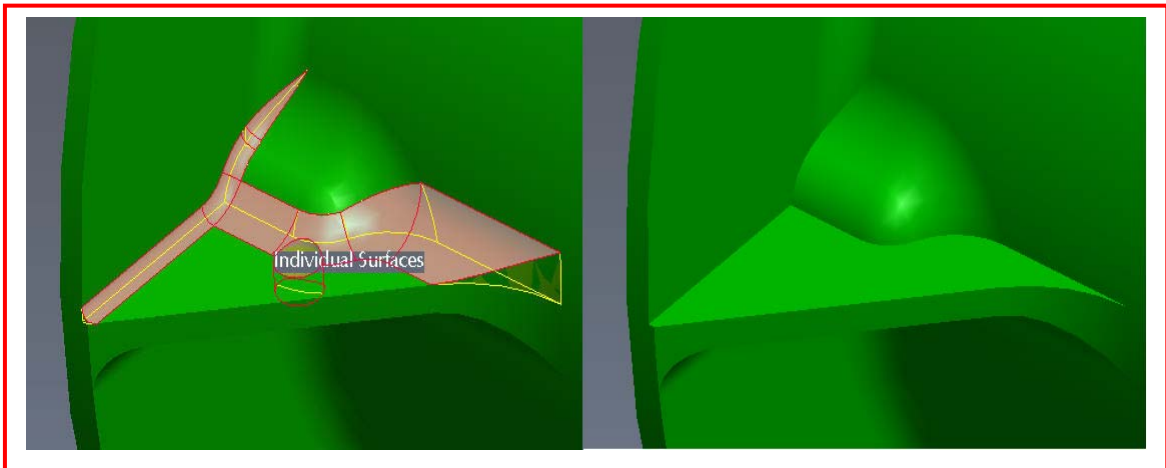
CONVERSION TOOLS

For more information on using the new Import Data Doctor for repairing models and making changes to the import data using the new Featurize Mode, you can follow the “Getting Started Guide” tutorial that can be launched from within Pro/ENGINEER.

Start the Help Center then choose Tutorials from the Quick Links section. Then select the link [“Getting Started with Import DataDoctor”](#).

2) New Remove Feature

A new feature called the Remove Feature was introduced in Pro/ENGINEER Wildfire 4. This is basically the same functionality that is being used inside the Import Data Doctor for Close, but has been extended for use in the modeling environment as a regular feature. So even if you are working on an imported model or a native Pro/E model, you can use the remove feature to “defeature” a part. For example, if you imported a part into Pro/ENGINEER and you want to eliminate some features to modify your design. You can simply select the surfaces that close up the feature and then select the Remove Feature Icon select Edit/Remove from the toolbar menu.



Adding a Remove Feature to a model by selecting the surfaces of a round and surfaces of a hole, then choosing the Feature.

Note that the Remove feature is an ordinary Pro/ENGINEER feature that will show up in the model tree, can be deleted, suppress or redefined just like any other Pro/E feature. In the above example, the remove feature that eliminates the holes and round could be deleted to get the geometry back!

3) Feature Recognition Technology

The third item that can be applied to make changes to imported geometry is the use of the new Feature Recognition Technology (FRT). This is a free Toolkit download in Wildfire 4.0 and can be downloaded from PTC’s website [at http://www.ptc.com/community/free-downloads.htm](http://www.ptc.com/community/free-downloads.htm).

This plug-in application for Pro/ENGINEER Wildfire 4.0 allows users to replace static geometries in imported 3D Boundary Representation models (IGES, STEP, etc) with parametric features. This plug-in is currently available for 32 bit and 64 bit Windows platforms only.

Its current capabilities are as follows to identify and replace static geometry with fully parametric content:

- Simple or Sketched Holes on Flat or Curved Surfaces
- Protrusions or Pockets on Flat or Curved Surfaces
- Extruded Slots
- Constant Radius Rounds
- Chamfers
- Table Pattern for Holes

Download it and try it today!

That's it! You just need to repeat the naming steps for each of the states you have created. Don't forget, Wildfire now allows you to create a representation that combines all of the options found in the View Manager.

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Tips of the Month

How to Bring Legacy Graphics Into an Interactive World

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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

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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/>

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Upcoming Events & Training Class Schedules

Upcoming, 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.

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