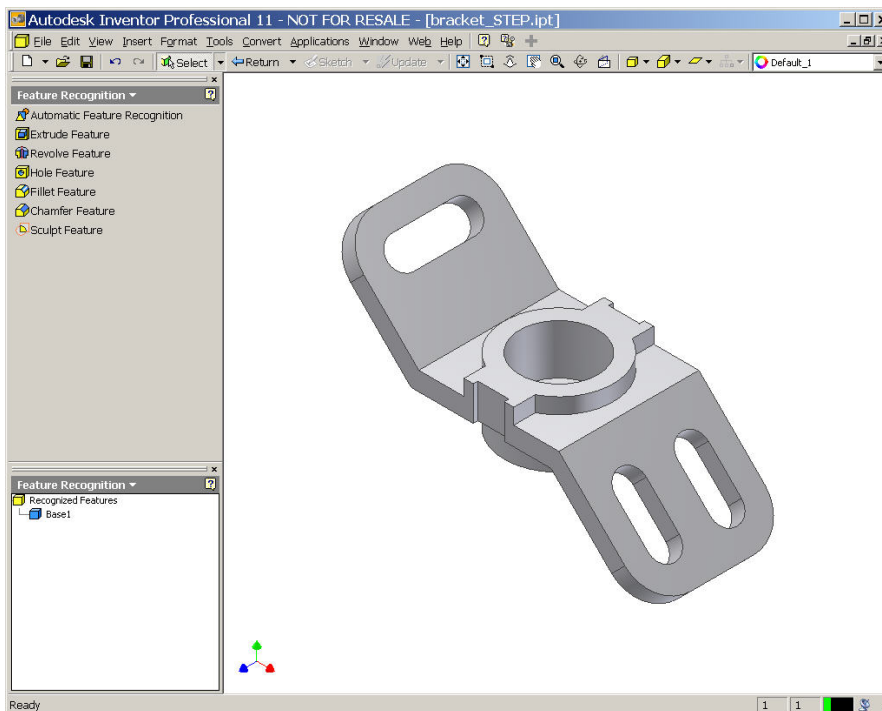


Feature Recognition for Inventor 11



This Autodesk Inventor® Release 11 add-in allows users to convert neutral 3D data files (STEP, SAT or IGES Solids) into fully featured Inventor® models. The application finds and creates hole features, extrude features, revolve features, fillets, and chamfers on a base body, either automatically, or with user guidance. With user guidance, precise design intent is assured. Once the features are identified, they are fully editable, acting as any native Inventor object would. This functionality is especially useful for Autodesk Inventor® customers who need to modify content that was not created in Inventor® originally. This add-in may be applied to Inventor® 11 DWF Extension as well.

Overview

After installing the application, the user will have the option to enter a feature mapping environment any time a STEP, SAT, or IGES Solid part file is opened.

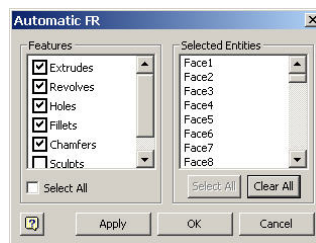
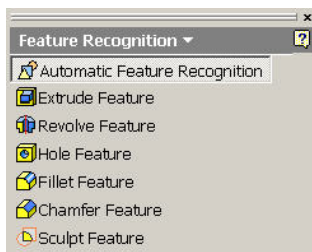


By selecting yes to this prompt, the feature recognition environment is presented to the user. In the feature recognition, user will convert the base body to a set of proposed Inventor features. In this technology preview, the features that can be created from base body geometry are limited to;

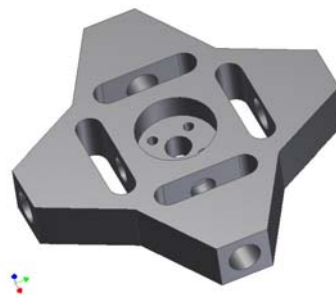
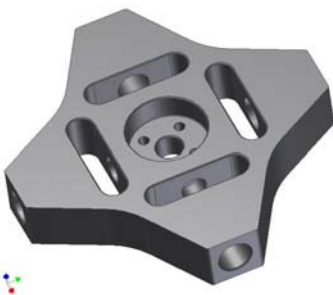
- Fillets
- Chamfers
- Holes
- Extrude
- Fillet
- Sculpt.

Automatic or Interactive Feature Mapping

Users may use automatic feature recognition, in which Inventor will find all features, or all features of a selected class. This allows users to control the feature order.



In general, users receive the best results by recognizing smaller features like fillets and chamfers first. The base body is transformed during the recognition process. Below is an example of an initial base body and the representation after all fillets and chamfers have been recognized.



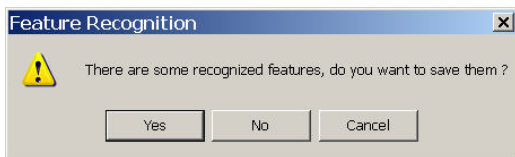
Notice how the base body is modified as the recognized features are identified.

Users may also use interactive mapping tools to identify features if the results of the automated mapping aren't as expected or do not follow design intent. Each interactive tool prompts for geometry selections that help identify the portions of the base body geometry to be created by the feature.

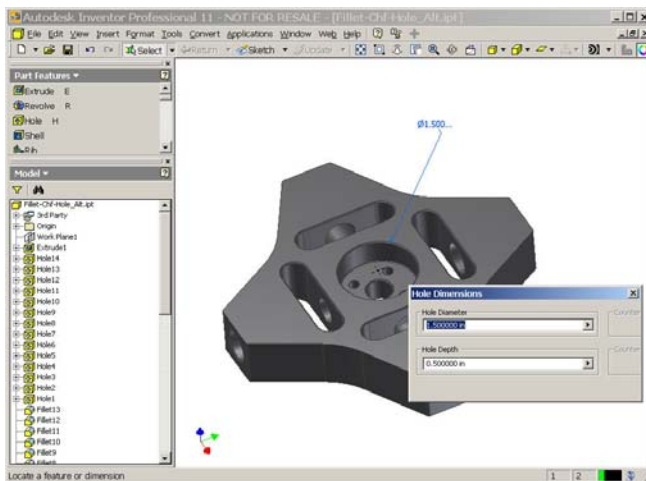
It is not a requirement to map all the features. If users only need to create Inventor holes to allow resizing or repositioning, that is allowed. The base body will include all geometry exclusive of the mapped holes.

Creating Inventor Features

Once the identification of features is completed, simply select the Inventor® Return command to exit the feature mapping environment. Inventor® will prompt to add the identified features to the model.



Once completed, a featurized Inventor model will be available for editing. In this example, it's easy to change the central hole diameter or depth.



That's all there is to it. It's an intuitive application. Monitor the shape of the base body as features are identified, if the resultant feature set is not to your liking, try a different order of mapping to get different geometry conditions.