



CAD Access Enhancements

- Unigraphics-MSC/PATRAN Interoperability
 - CATIA Access Improvements
 - MSC/PATRAN ACIS Access
 - CAD Direct Access Support

2.1 Unigraphics-MSC/PATRAN Interoperability

The following new features in MSC/PATRAN Version 8 provide significant improvements in Unigraphics interoperability:

- Features and Parameters
- Assembly and Preview Access
- Attribute Automation

The following sections describe these features in detail.

Features and Parameters

MSC/PATRAN Version 8 introduces a breakthrough in CAD/CAE interoperability which dramatically reduces the analysis modeling time required for design modification. **Figure 2-1** gives an overview of the process.

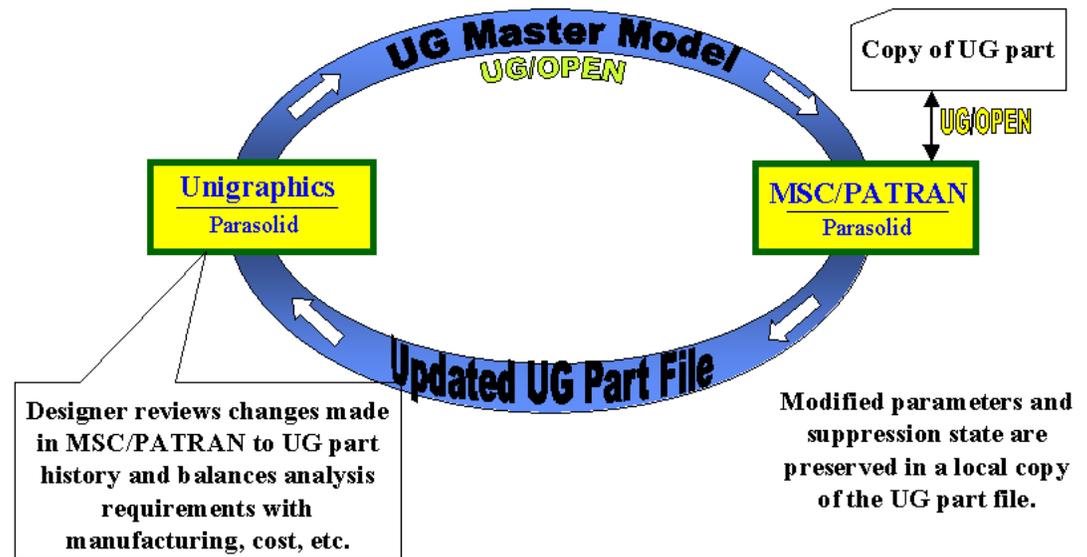


Figure 2-1 Unigraphics-MSC/PATRAN Interoperability Process

MSC/PATRAN Driven Feature Modification

Unigraphics features can now be accessed in MSC/PATRAN for suppression, unsuppression, and parameter modification, reducing or eliminating the time required to manually update the analysis model.

When a change is made to a Unigraphics feature (i.e. modifying a hole diameter or suppressing the hole) within MSC/PATRAN, the geometry is updated to reflect the change. MSC/PATRAN preserves the mesh control parameters (mesh seeds, curvature based settings), mesh size, loads and boundary conditions (LBCs),

properties (materials, thickness, etc.) and re-applies these parameters to the modified geometry. Modifying, suppressing, or unsuppressing a feature is accessed using the Edit/Feature menus under Geometry.

**Unigraphics Driven
Feature Modification**

In addition to driving the CAD features from within MSC/PATRAN, a change can be made to the Unigraphics master model and automatically updated by re-importing into the same database. MSC/PATRAN recognizes if the same part file is re-imported and prompts the user with a form stating that this model already exists in the database, “Do you wish to update the features?”. If the response is “YES”, then the features’ parameters are updated but the suppression state of the MSC/PATRAN model is preserved instead of the CAD model suppression state. Many times, the CAE model has many more features suppressed to allow for an efficient analysis mesh.

**Introduction to
Unigraphics Features**

Feature based solid modeling uses features, such as holes, slots, and bosses, in a model. You can edit the dimensions and locations of these features. For example, a hole is defined by its diameter and length. You can edit any of these parameters by entering new values to update the geometry and CAE attributes. The position of the hole can also be modified relative to a reference location on the model.

The following table describes Unigraphics features.

Type	Feature Name	Definition
Sketched Features	Sketch	Used to create two-dimensional representations of profiles associated with the part. In most cases, the sketch is extruded or revolved to create a solid or sheet body. It is good practice to name the sketch parameters clearly since a sketch feature may have many parameters. Otherwise, it is difficult to know which parameter to modify for a desired change.
Form Features	Hole	Removes material in the shape of several types of standard holes.
	Slot	Removes material in the shape of a straight slot with rounded ends or completely through the two faces.
	Groove	Removes material in the shape of a groove, as if a tool moved inward (or outward) on a rotating part. The target body must be cylindrical.
	Pocket	Removes rectangular, cylindrical, or general shape if made from curves.
	Boss	Adds material in a cylindrical or conical shape.
	Pad	Adds material to a solid body.
	User-Defined	Custom designed feature.
Swept Features	Extruded Body	Create a body by sweeping the section curves a linear distance in a specified direction.
	Body of Revolution	Create a body by revolving the section curves about an axis.
	Sweep Along Guide	Create a body by extruding an open or closed boundary sketch, curve, edge or face along a guide (path) formed by one or a series of curves, edges or faces.
Reference Features	Datum Plane	Used as an aid when planes are not available. Datum planes are used in creating other features.
	Datum Axis	Used to create a reference axis, which can be used to create datum planes, revolved features, extruded bodies, etc.

Type	Feature Name	Definition
Primitive Features	Block	A block specified by either a) Edge Lengths, Corner, b) Height, Two Points, c) Two Diagonal Points.
	Cylinder	A cylinder specified by either a) Diameter, Height, b) Height, Arc.
	Cone	A cone specified by either a) Diameters, Height, b) Diameters, Half Angle, c) Base Diameter, Height, Half Angle, d) Top Diameter, Height, Half Angle, e) Two Coaxial Arcs.
	Sphere	A sphere specified by either a) Diameter, Center, b) Select Arc.
	Tube	A tube specified by a cross-section diameter and a guide string.

Type	Feature Name	Definition
Feature Operation	Unite	Combines multiple solid bodies into one.
	Subtract	Subtracts one or more solid bodies from another
	Intersect	Creates solid bodies from material that is common to two solid bodies.
	Trim Body	Trims off part of a body using a plane, sheet body, etc.
	Offset Faces	Offsets one or more faces a specified distance.
	Sew	A complex solid by sewing a series of connected solids or sheets together.
	Extract	A body (face region of a body, or an entire body) extracted from another body.
	Sheets from Curves	The simplest form sheet body between two curves.
	Split Body	Divides one or more bodies into separate bodies
	Patch Body	Patches a sheet to a sheet or solid body, removing faces as needed.
	Simplify Body	Removes connected sets of faces from a solid body while retaining the ability to retrieve them.
	Hollow	Hollows out or creates a shell around a single solid body.
	Blend	Adds or removes edges from a solid body by rounding.
	Chamfer	Adds or removes material from the edges of a solid body by beveling.
	Taper	Applies a draft angle to selected faces of a solid body.
	Instance	Shape linked instances of features. For example, similar holes could be instances of another hole feature.
Thread	Parametric thread on a cylindrical face.	
Scale	Increases or decreases the size of the solid or sheet body.	

Architecture When features are modified, a small subset of Unigraphics performs the changes to update the geometry in the MSC/PATRAN database as well as to the UG part file. To avoid modifying the original UG model, a local copy of the part file is made in a newly created directory by PATRAN. The directory is created in the same directory where PATRAN was started with the copied part file named:

```
database_name_ug_copy/ug_part_name.prt_msc_database_name
```

where `database_name` is the name of the MSC/PATRAN database and `ug_part_name.prt` is the name of the original UG part file. If this part file is not in the expected location (i.e. the files are moved to a different workstation) when an update is initiated, the user will be prompted for the new location of the part file. If the part file is not available, additional modeling can continue with the PATRAN model, but no feature modification will be allowed until the copy is found or the master model is re-imported and synchronized with the PATRAN database (master model feature state with PATRAN suppression state).

This local copy of the part file is maintained in the same feature state as the geometry in the PATRAN database. These feature modifications and suppressions are performed using Unigraphics native UG/OPEN toolkit which has the benefit of supporting advanced feature nesting. For example, a modified gear tooth pitch might drive the number of adjacent holes. **Figure 2-2** and **Figure 2-3** demonstrate this process.

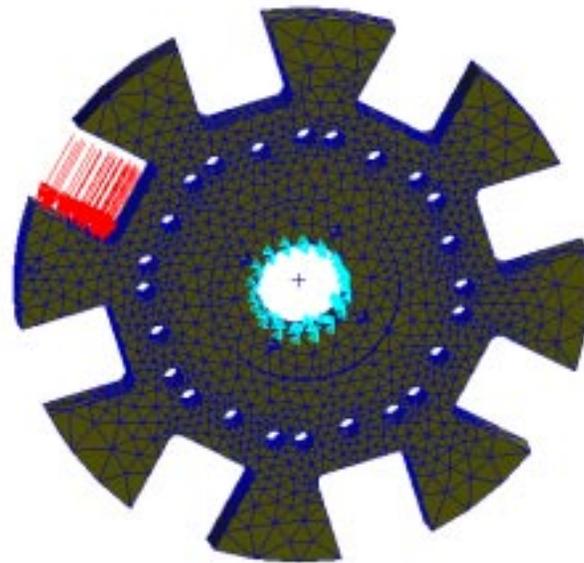


Figure 2-2 Initial Design with 8 Slots

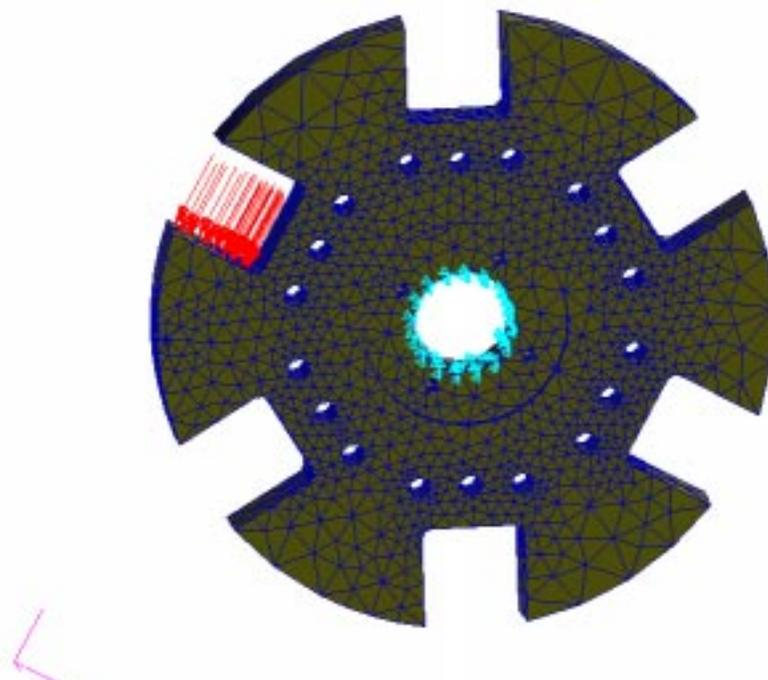


Figure 2-3 Parametric Update with Automatic Remesh and Preservation of Loads

Tips and Limitations

1. Only parameterized Unigraphics parts can be modified. Some UG models contain recognizable features (holes, blends, etc.) but are not built parametrically. These features cannot be modified using the new capability.
2. Modification of features are only supported for Unigraphics parts. Parts with multiple solids are supported in a single part file, but assembly feature modification is not supported.
3. Only the features from the first parameterized part file are supported in a single PATRAN database. Additional part files and/or databases with existing parameterized part files may be imported, but the feature list will be excluded.
4. Upgraded MSC/PATRAN databases from previous versions will not contain features.
5. Import of an MSC/PATRAN database into another MSC/PATRAN database will not include the feature list.
6. MSC/PATRAN supports modifications that Unigraphics can perform with direct UG/OPEN calls. Feature modifications that cannot be performed in interactive Unigraphics will also fail in MSC/PATRAN with an error message stating that “The Unigraphics model was unable to be updated with the feature suppression states and parameter values in this database...”.

7. Some modifications that require additional user intervention in interactive Unigraphics, such as reordering features, are not supported in MSC/PATRAN.
8. If a part file has entities that will not be used during the parametric studies, it is recommended to deselect the entity (e.g. Wire Body) from the Unigraphics Import Options subform before import. Every subsequent suppression will ignore the unselected entity type.
9. Mesh control parameters such as mesh seeds and face seeds (meshing a face of a solid before solid meshing) are preserved if the update does not cause Unigraphics to change the internal entity id. Hard points and curves are not supported during update.
10. Entities that change topologically (such as a face with a different number of vertices after an update) will lose their CAE properties. The single exception is face pressure, which will be preserved assuming that Unigraphics returns the same id for the face.
11. The feature list displayed in MSC/PATRAN will in some cases be represented differently than how the features are listed within Unigraphics. For example, MSC/PATRAN might show a CIRCULAR_ARRAY of holes with the number of holes modifiable. Unigraphics might display this same list as individual INSTANCES of a SIMPLE_HOLE.
12. Any geometry and its associated CAE attributes created from (referencing) the UG geometry will be deleted upon update of a feature. A warning form will prompt the user before this occurs.
13. Neither a Unigraphics license nor installation is required. Only the Unigraphics selection of the MSC/PATRAN installation is required.

Assembly Preview and Access

MSC/PATRAN Version 8 extends Unigraphics access to support assemblies and assembly occurrences filtering. Frequently, an analysis is only required on a subset of a large assembly. Users have the option to preview a list of all of the assembly components for a faster import of only the required geometry.

For Parasolid transmit files of assemblies created by products other than Unigraphics, only one instance of the total number of instances existing in the assembly will be imported.

Attribute Automation

Access of Unigraphics name and object attributes is supported in Version 8, allowing for automation of redundant tasks. Both Named and Object attributes are supported. Unigraphics models can be tagged with customer specific attributes for such tasks as bolt or weld point locations, with automated customer specific modeling techniques. For example, a curve or surface in Unigraphics can have a user specified attribute giving the name of this surface (e.g. "Upper Bulkhead Connector"). PCL can be written to find the surface with this named attribute in MSC/PATRAN, and applying a particular mesh, mesh size, load, or attachment technique. Applications such as these create a significant opportunity for reducing the analysis modeling time.

2.2 CATIA Access Improvements

Access Methods MSC/PATRAN Version 8 includes major enhancements in CATIA integration. You can now access CATIA models directly from the MSC/PATRAN File/Import form.

The new Direct CATIA Access capability requires a new license. Contact your sales representative for details. A CATIA license is required to access the CATIA model from MSC/PATRAN.

Previously, CATIA geometry could only be accessed by:

1. Exporting the CATXPRES intermediate file (.cat suffix) from the CATIA application.
2. Importing the intermediate file using the MSC/PATRAN user interface.

The Direct Access method and the CATXPRES method are both supported in Version 8.

In environments where CATIA and MSC/PATRAN are on separate networks, an intermediate file can be exported from a workstation on the CAD network to send to the analysis network. Copy the following executable from the MSC/PATRAN installation and create the express file by entering the command at the UNIX prompt:

```
% p3_catia_express catia_model_name.model -fsp/model_directory_path -t
```

where `catia_model_name.model` is the name of the CATIA model and `model_directory_path` is the path to the CATIA model. The `-t` option specifies an ASCII express neutral file (*.exp) type. Otherwise a binary (*.bxp) file will be generated. Both types of files can be imported into MSC/PATRAN using File/Import/Express/Neutral File. The layer and entity filtering options are available from this form.

Filtering CAD models frequently contain more detail than is required for analysis purposes. MSC/PATRAN now provides two types of filters to enable you to import only the relevant portions of the CATIA model for analysis. The types of filters are:

- entity filter
- layer filter

Filtering can save you the effort of manually selecting and deleting unnecessary geometry from your analysis model, resulting in:

- simplified access to specific information.
- reduced analysis preparation time.

These filters are available from the File/Import/CATIA Options form.

Entity filtering limits the types of entities that are imported into MSC/PATRAN. For example, if your analysis only requires surface geometry and your model contains curves or solids as well, you can use the entity filter to select only surface entities for importing.

Layer filtering imports only the portions of the model specified by layer ID. Layer IDs must be specified on the CAD model prior to importing.

Grouping Layers

You can also group layers of the model together when importing geometry. For example, a large car assembly model might be grouped so that the body and its associated trim can be analyzed independently. You could create a group called “body” containing layers 3, 4, 5, and 6, and create another group called “trim body” with layers 23, 24, 25, and 26. You can then import the “body” and “trim body” groups and ignore all other layers.

2.3 MSC/PATRAN ACIS Access

Geometry models from CAD systems based on the ACIS geometry kernel can be imported directly into MSC/PATRAN. The new ACIS access capability requires a new license. Contact your sales representative for details.

CAD model files in the .sat export form are used to access the geometry. MSC/PATRAN will create a Parasolid transmit (.xmt_txt) file from the input .sat file and import it through the Parasolid import function.

ACIS access in MSC/PATRAN is based on the ACIS 2.1/3.0 Geometric Modeler. This new access capability provides a path within the MSC/PATRAN analysis environment to a wide variety of ACIS-based CAD systems and ACIS-Compatible applications, including AutoCAD Mechanical Desktop, Bravo, CADKEY, IronCad, MSC/ARIES, SolidDesigner, TurboCAD, and Vellum. A summary of features and limitations follows.

Features

- Provides high quality exchange of geometric and topological data between ACIS and Parasolid geometry kernels.
- Supports scaling of ACIS models for unit conversion.
- Supports creation of tolerant Parasolid models whenever required.

Limitations

- Wireframe and non-manifold geometry are not supported.
- ACIS and Parasolid units do not match. The default unit used in Parasolid is meters and the default unit used in ACIS is millimeters. End applications based on these kernels may have different default units. To achieve consistency, ACIS models must be scaled after conversion. Scaling a model typically does not cause any problems, so the conversion from ACIS to Parasolid is reliable.

2.4 CAD Direct Access Support

MSC/PATRAN Version 8 supports the following versions of CAD software.

	HP		SGI		Sun		IBM RS/6000		Digital		NT	
	V7.5	V8	V7.5	V8.0	V7.5	V8.0	V7.5	V8.0	V7.5	V8.0	V7.5	V8.0
CADDS	7.0	8.0	7.0	8.0	7.0	8.0						
CATIA (CATXPRES)	4.1.8	4.2.0	4.1.8	4.2.0	4.1.8	4.2.0	4.1.8	4.2.0	4.1.8*	4.2.0*	4.1.8*	4.2.0*
Direct CATIA Access		4.2.0		4.2.0		4.2.0		4.2.0		4.2.0*		4.2.0*
EUCLID 3	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2	3.1.2
Pro/ENGINEER	18.0	19.0	18.0	19.0	18.0	19.0	18.0	19.0	18.0	19.0	18.0	19.0
Unigraphics	13.0	14.0	13.0	14.0	13.0	14.0	13.0	14.0	13.0	14.0	13.0	14.0

* Import only.