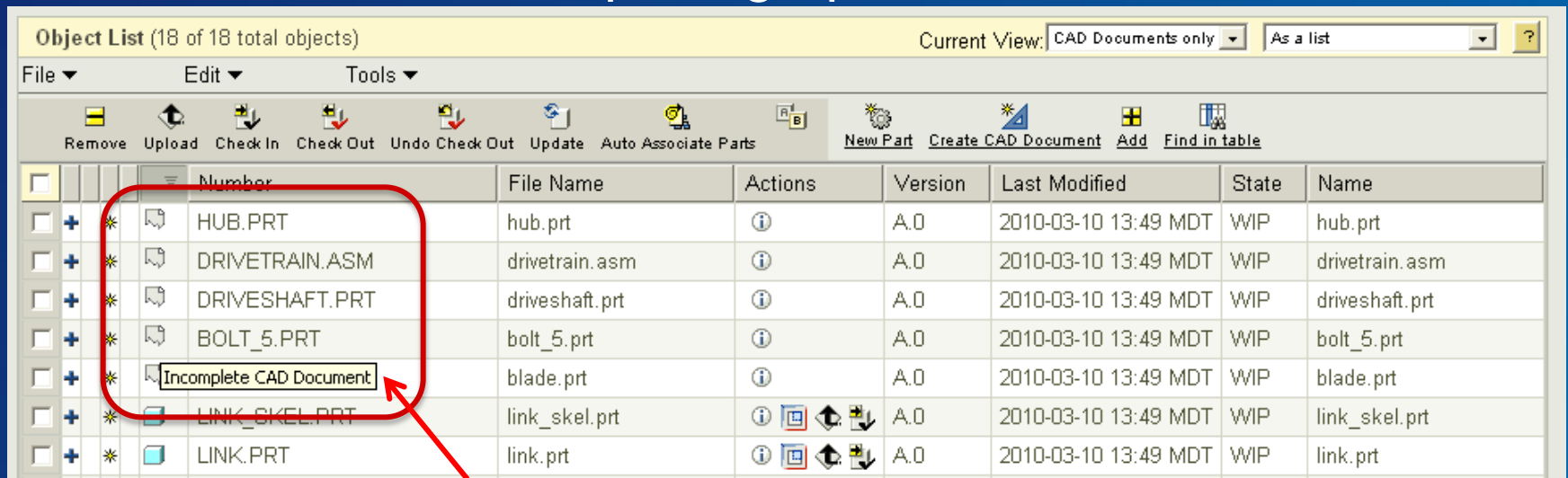


# PRELIMINARY INFORMATION



















- Presentation information and graphics are based on:
  - ProEngineer WildFire 4.0 M110
  - Windchill PDM 9.0
  - Incomplete CAD Documents (Ghost Objects) as shown in Workspace graphic below



Object List (18 of 18 total objects) Current View: CAD Documents only As a list

File Edit Tools

Remove Upload Check In Check Out Undo Check Out Update Auto Associate Parts New Part Create CAD Document Add Find in table

		Number	File Name	Actions	Version	Last Modified	State	Name
<input type="checkbox"/>	+	* 	HUB.PRT		A.0	2010-03-10 13:49 MDT	WIP	hub.prt
<input type="checkbox"/>	+	* 	DRIVETRAIN.ASM		A.0	2010-03-10 13:49 MDT	WIP	drivetrain.asm
<input type="checkbox"/>	+	* 	DRIVESHAF.T.PRT		A.0	2010-03-10 13:49 MDT	WIP	driveshaft.prt
<input type="checkbox"/>	+	* 	BOLT_5.PRT		A.0	2010-03-10 13:49 MDT	WIP	bolt_5.prt
<input type="checkbox"/>	+	*  Incomplete CAD Document	blade.prt		A.0	2010-03-10 13:49 MDT	WIP	blade.prt
<input type="checkbox"/>	+	* 	LINK_SKEL.PRT	  	A.0	2010-03-10 13:49 MDT	WIP	link_skel.prt
<input type="checkbox"/>	+	* 	LINK.PRT	  	A.0	2010-03-10 13:49 MDT	WIP	link.prt

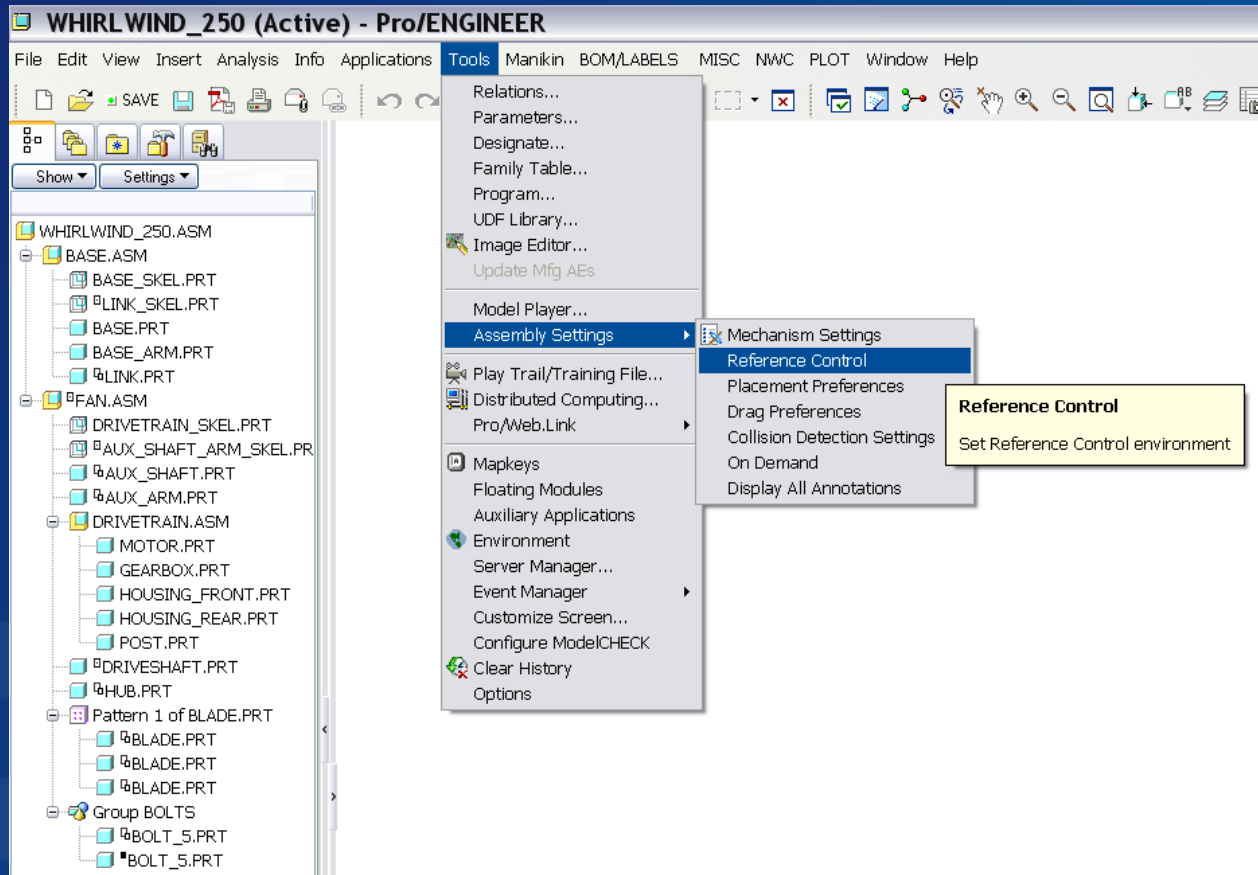
(Tool Tip when mouse pointer is hovered over icon)

# PRELIMINARY INFORMATION

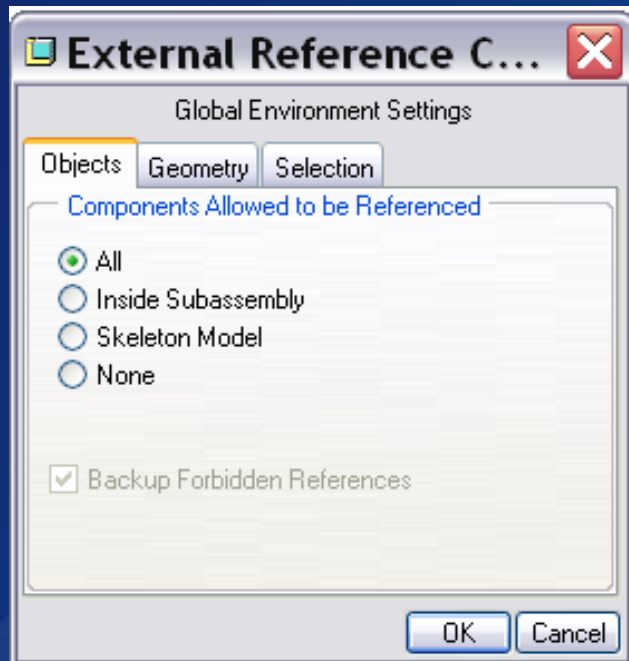
- What could cause a “Ghost” object?
- The list of commonly used Pro/ENGINEER features that utilize dependencies includes, but is not limited to:
  - Sketch References
  - Drawing cosmetic references
  - Layout-driven parameter references
  - Copy geom features
  - Cutout features
  - Merge features
  - Inheritance features
  - Component replace/restructure and interchange groups
  - Suppressed models

# CONFIGURING ASSEMBLY SETTINGS

- Tools>Assembly Settings>Reference Control



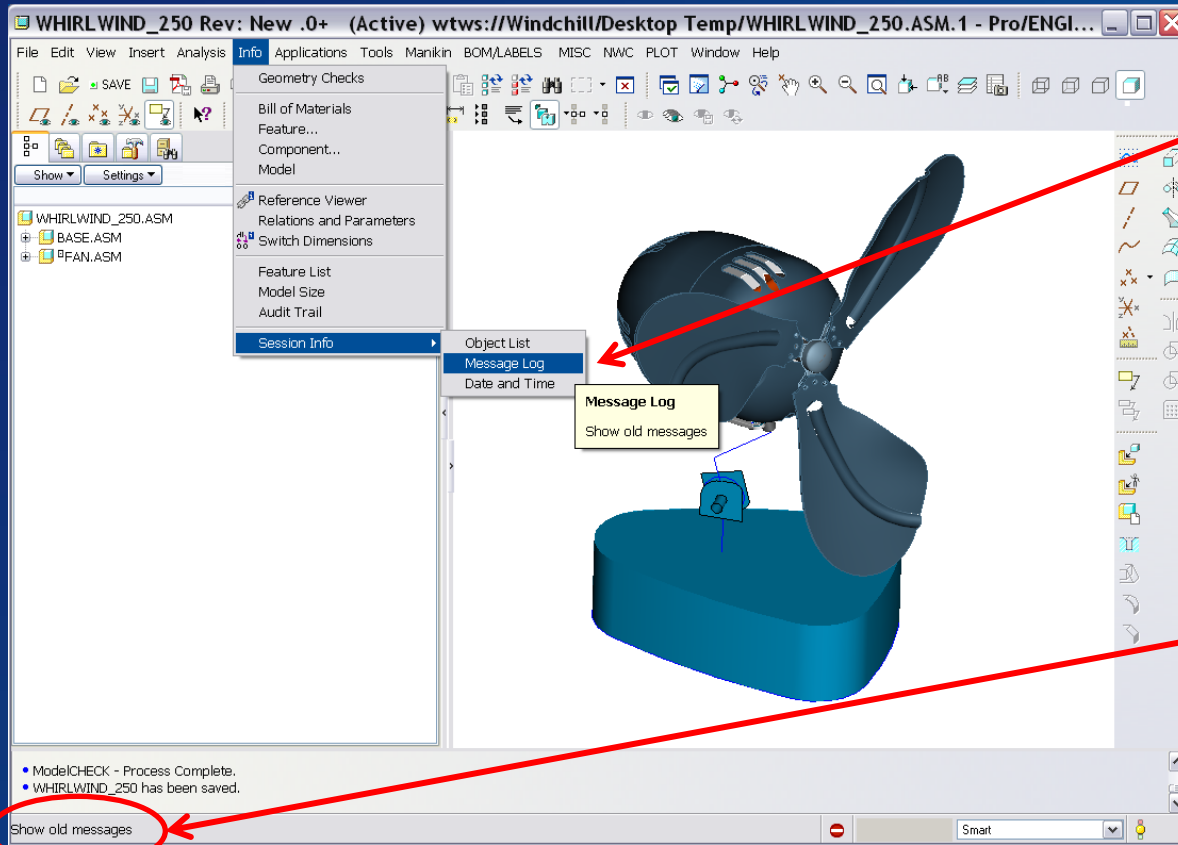
# CONFIGURING ASSEMBLY SETTINGS (cont)



- Use the External Reference Control dialog box to specify global environment settings to control scope, reference handling, reference selection, and color feedback during selection of references
- The “All” option allows External References but if the models are not kept together, the link is broken and “Ghost” objects will appear in your PDM Workspace.

# ProE MESSAGE LOG

- When a model is opened, ProE will inform you of many potential or undesirable issues. Review the Message Log for detailed information.

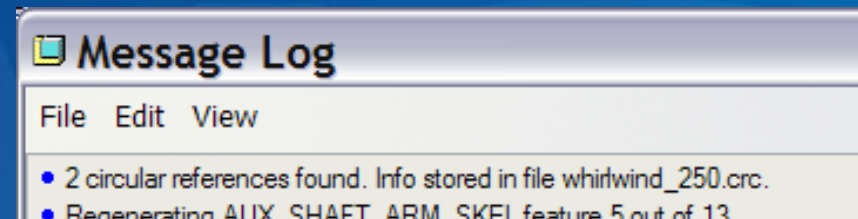
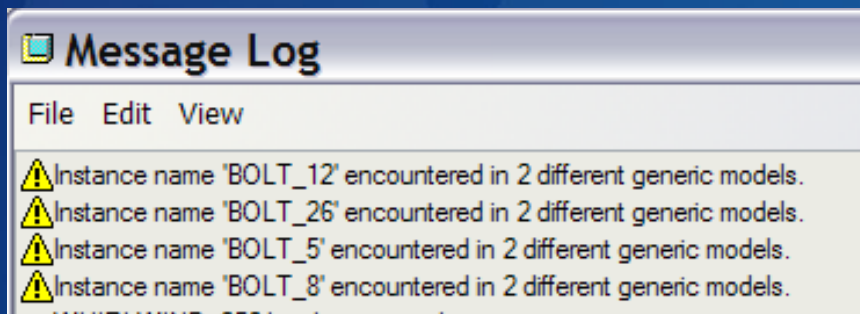
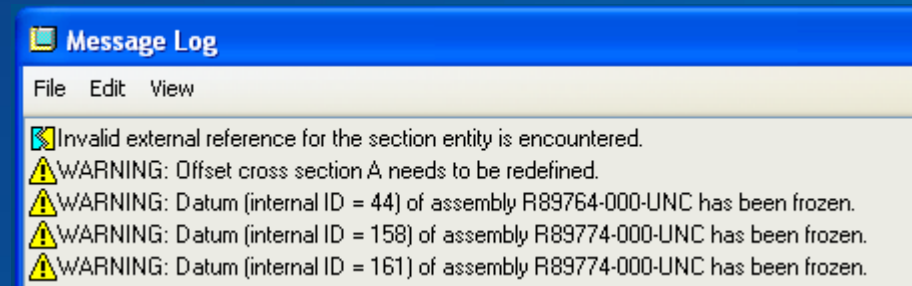
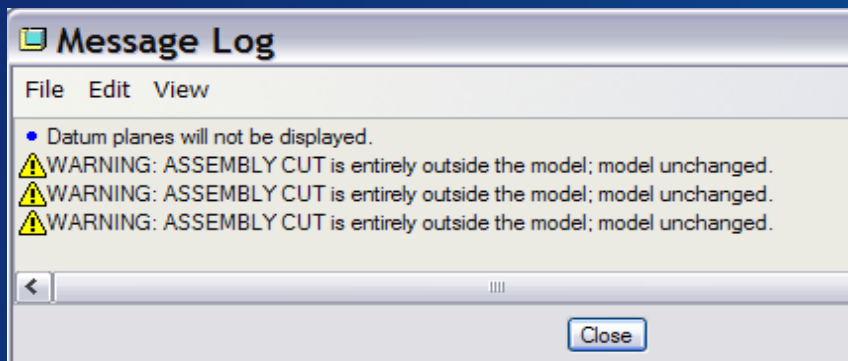


Open Message Log for complete list of warnings and information

Quick view of info that scrolls by as model is opened

# ProE MESSAGE LOG (cont)

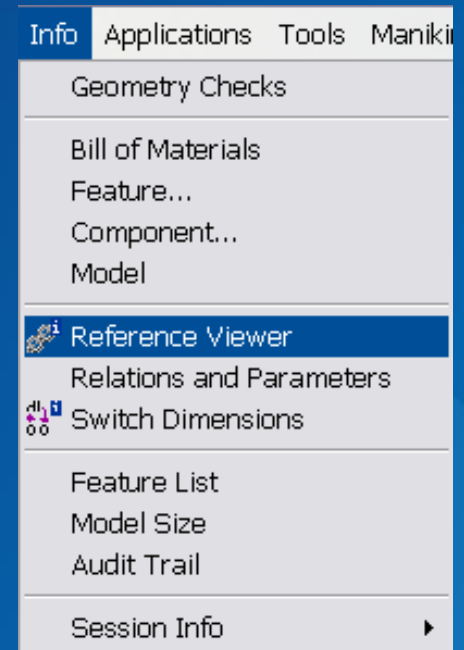
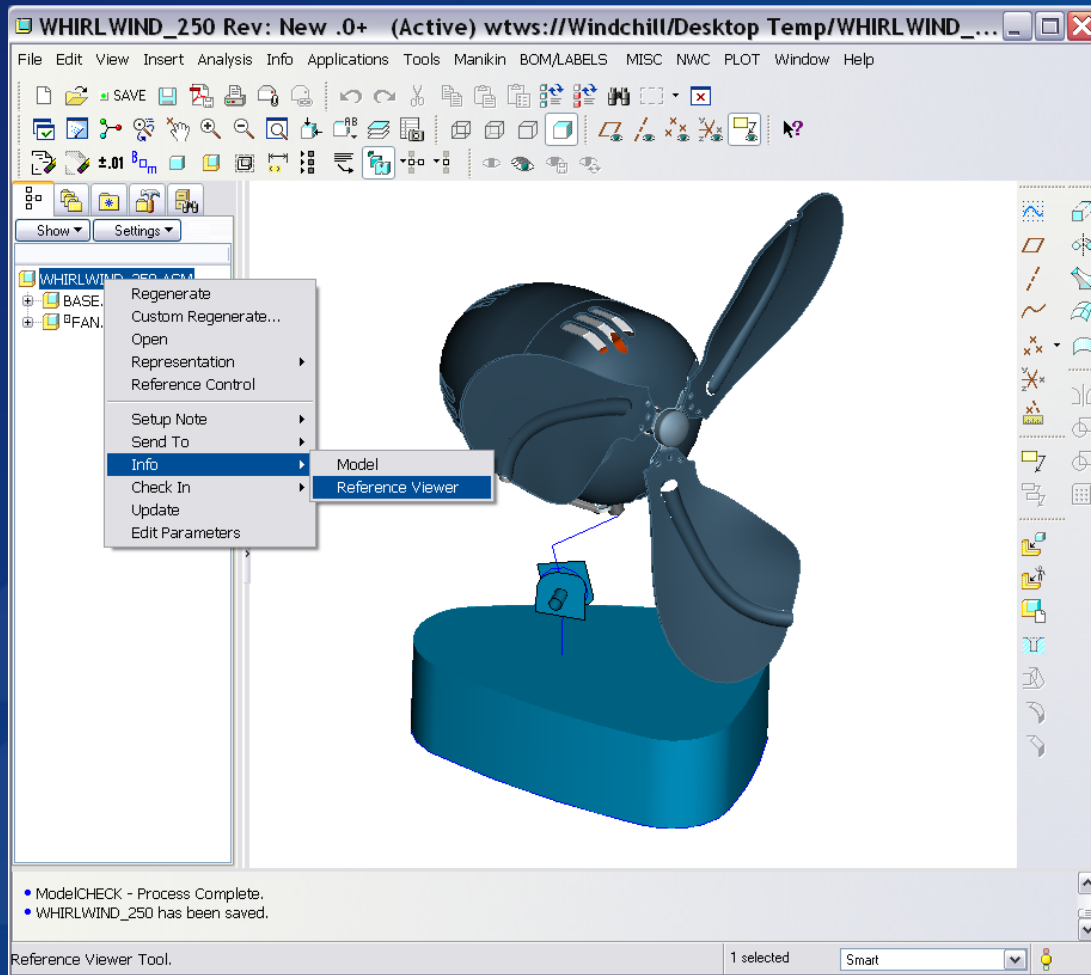
- Typical messages of undesired conditions
- Yellow Warning Signs should be investigated
- These errors may or may not identify missing references that would cause Ghost Objects
- Use the Reference Viewer to observe these items



# ProE REFERENCE VIEWER

• Right-click on model, select Info>Reference Viewer to investigate missing references and circular references

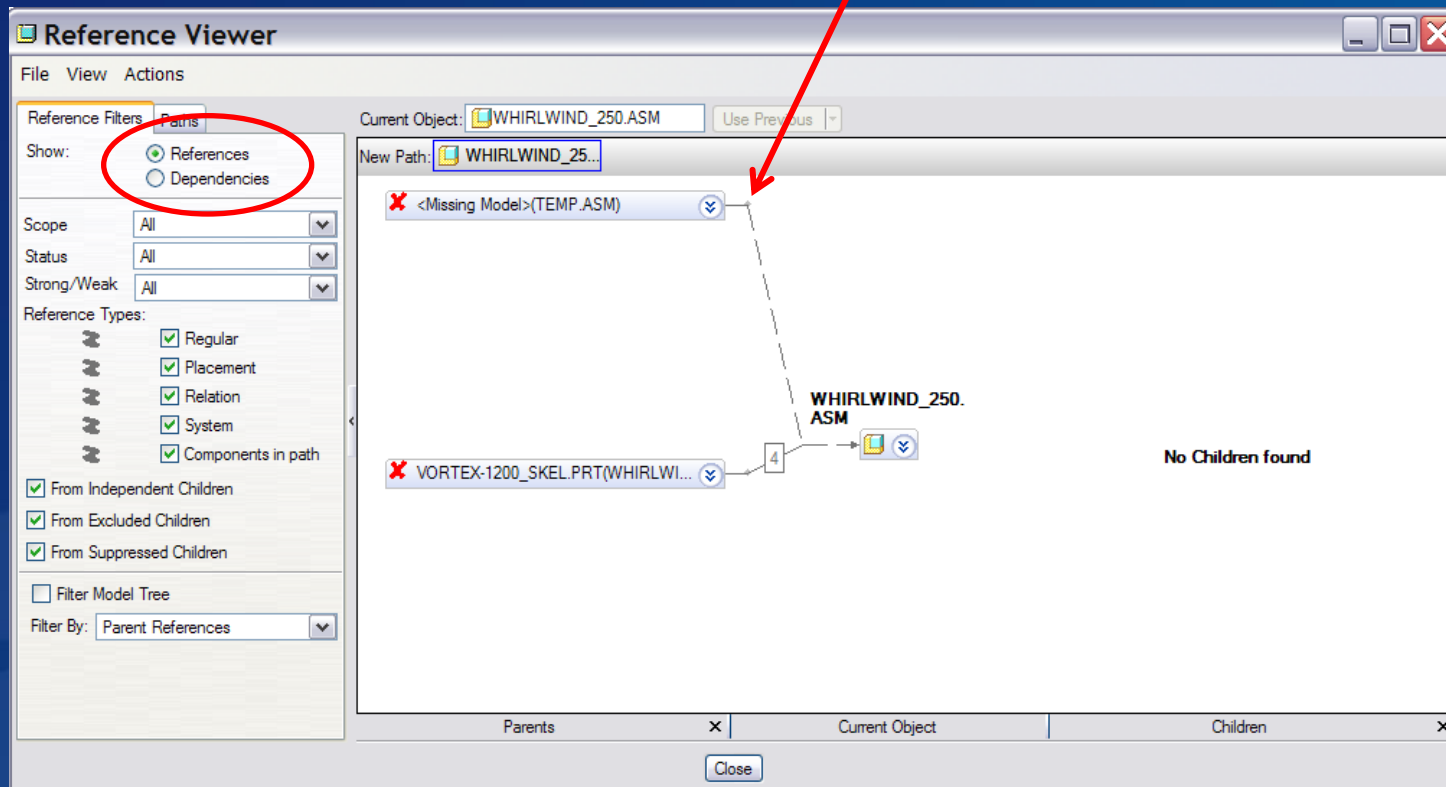
• Reference Viewer also found with Top Menu>Info Menu



# ProE REFERENCE VIEWER (cont)

- Two types of references can be selected
  - References
  - Dependencies

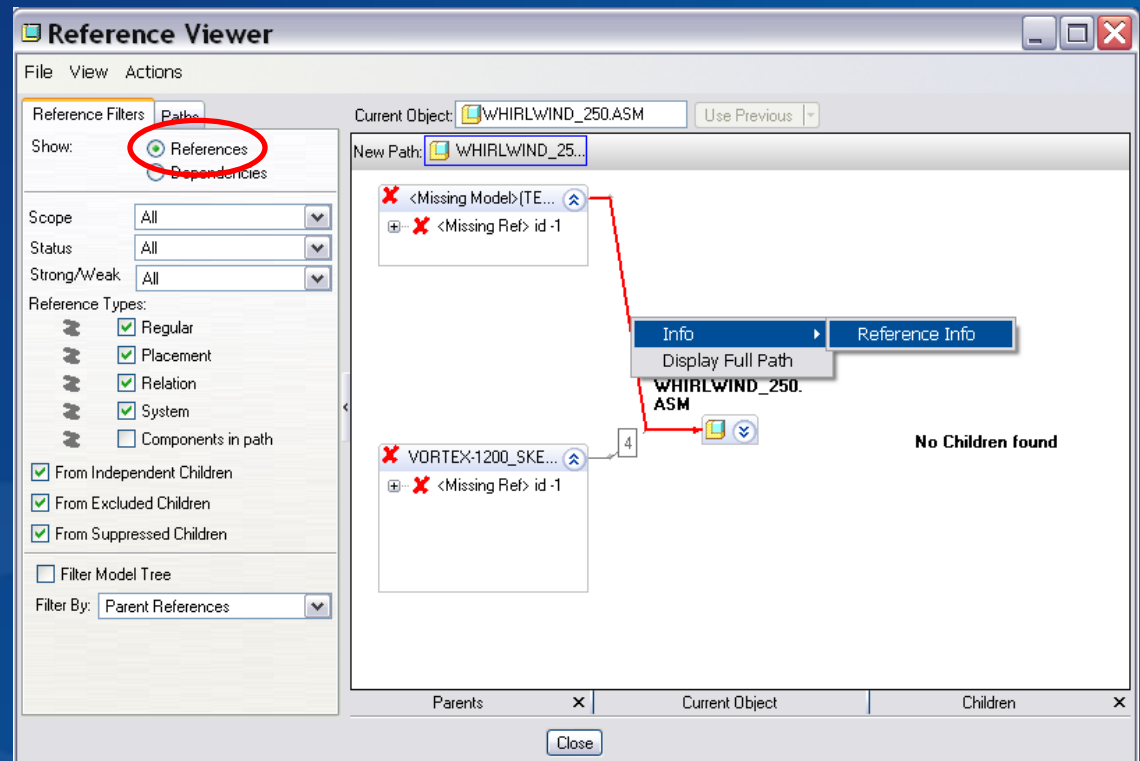
Gray Dot indicates external ref.





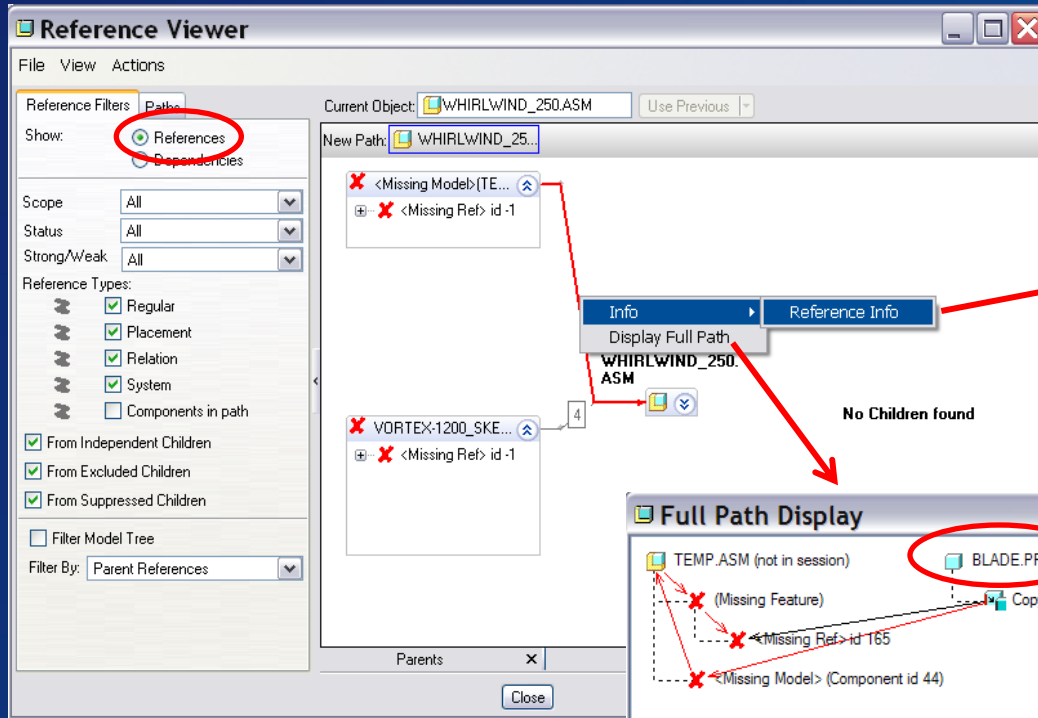
# ProE REFERENCE VIEWER (cont)

- Typical missing references
- Right-click on connecting line to bring up addition information
- Two options available to view details (see next slide)
  - Reference Info
  - Display Full Path

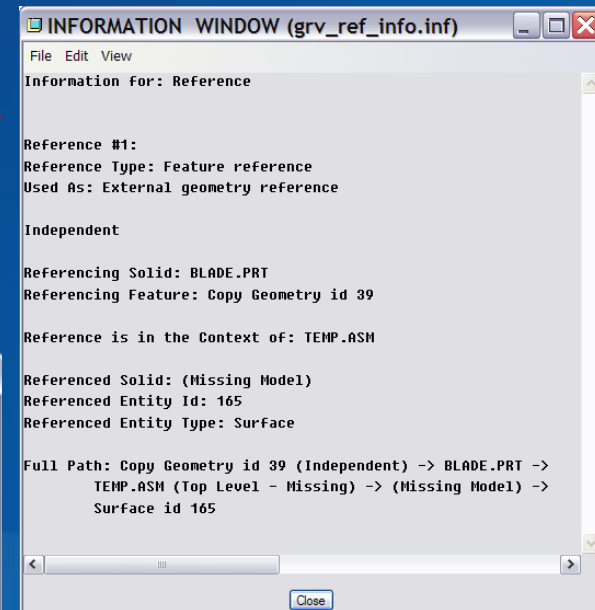


# ProE REFERENCE VIEWER (cont)

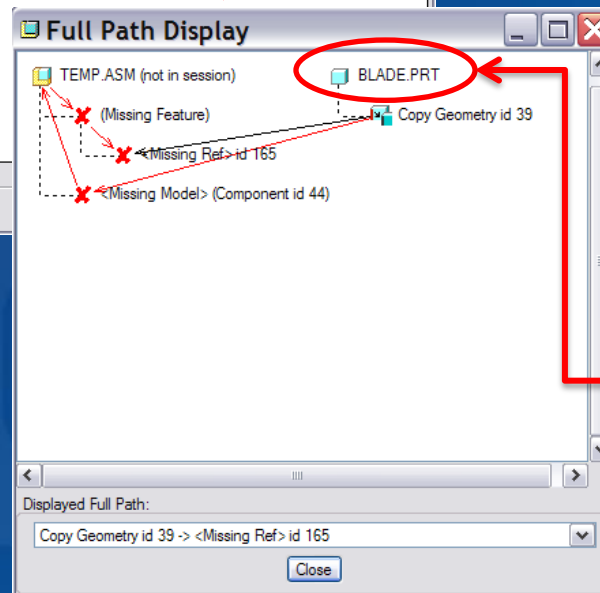
- Two menus, same information



Reference Info Display  
(text)



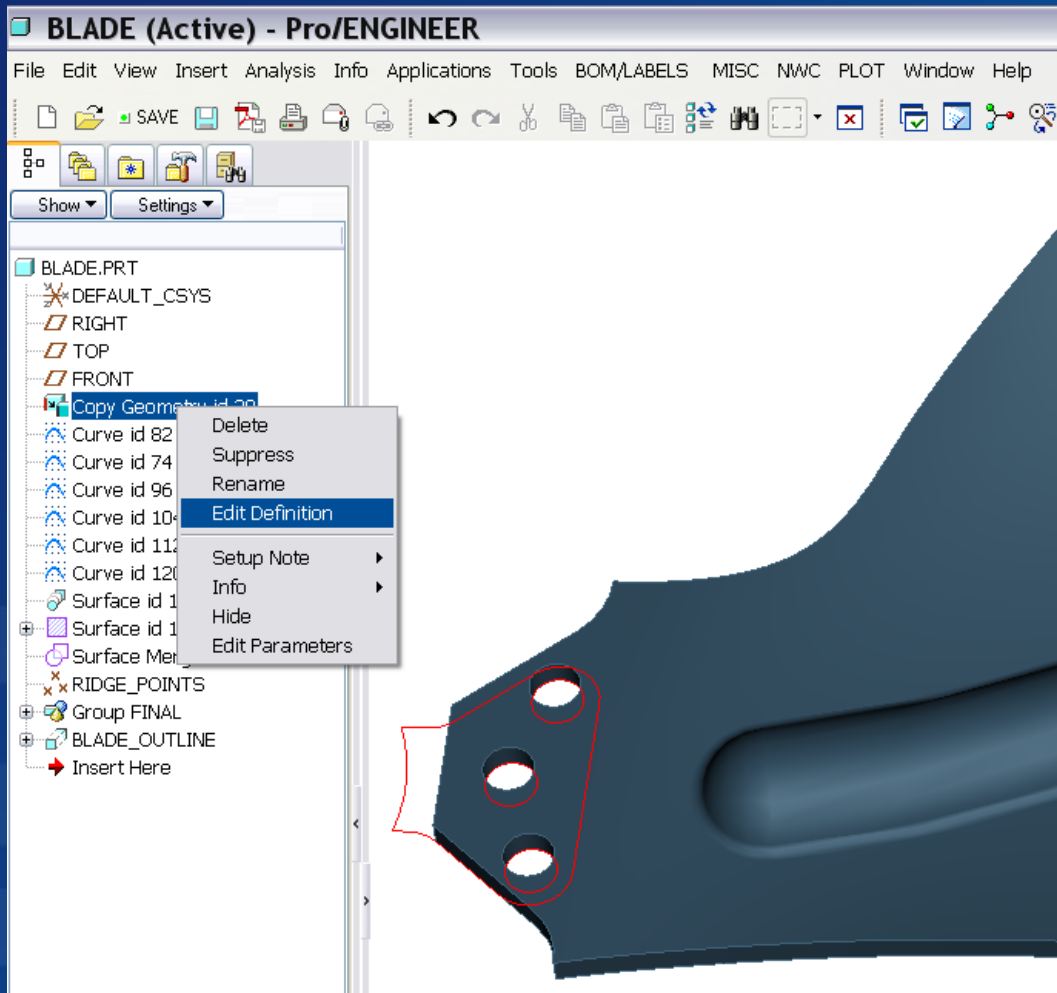
Full Path Display  
(graphical)



BLADE.PRT model  
is the culprit

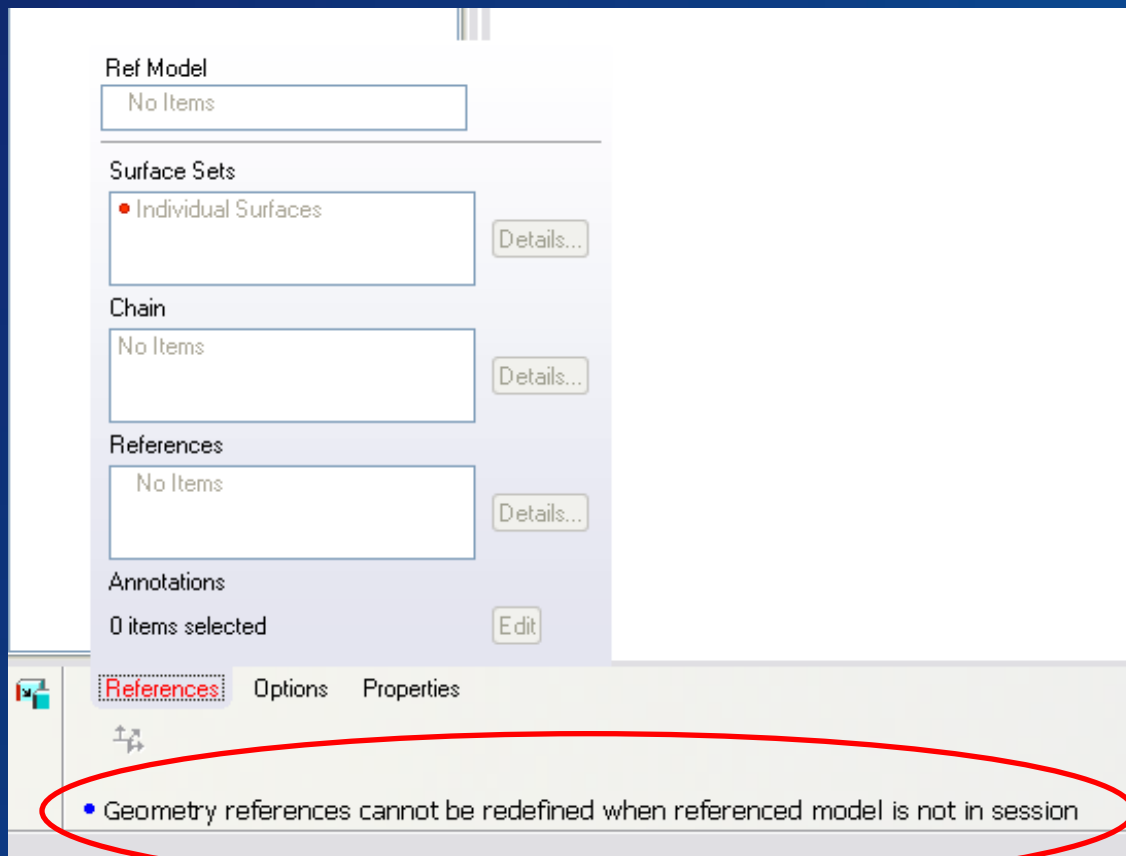
# ProE REFERENCE VIEWER (cont)

- Open the BLADE.PRT model to investigate
- Right-click on “Copy Geometry”, select Edit Definition



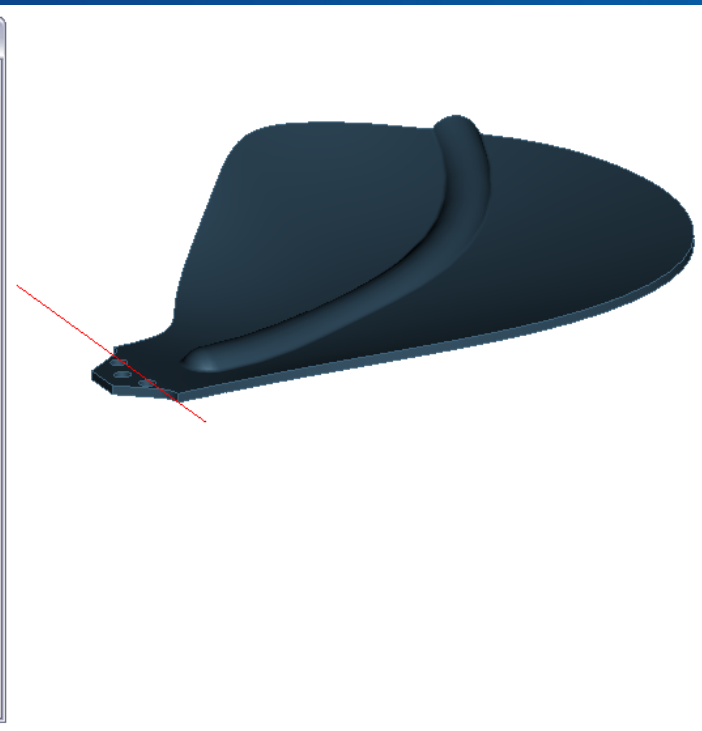
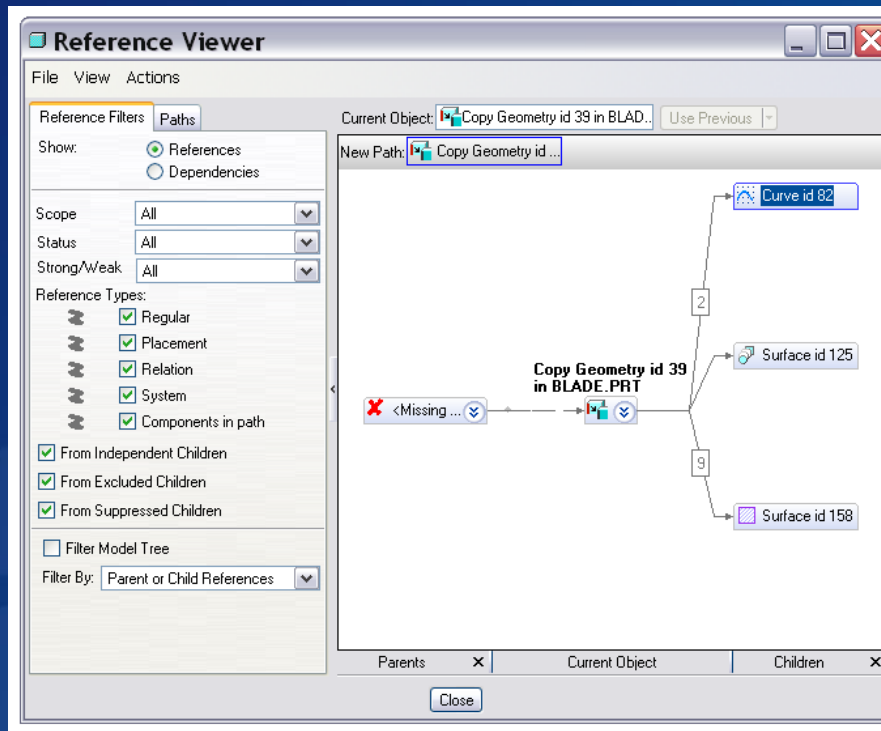
# ProE REFERENCE VIEWER (cont)

- Observe message window
- Without the missing model, the “Copy Geometry” feature references cannot be redefined



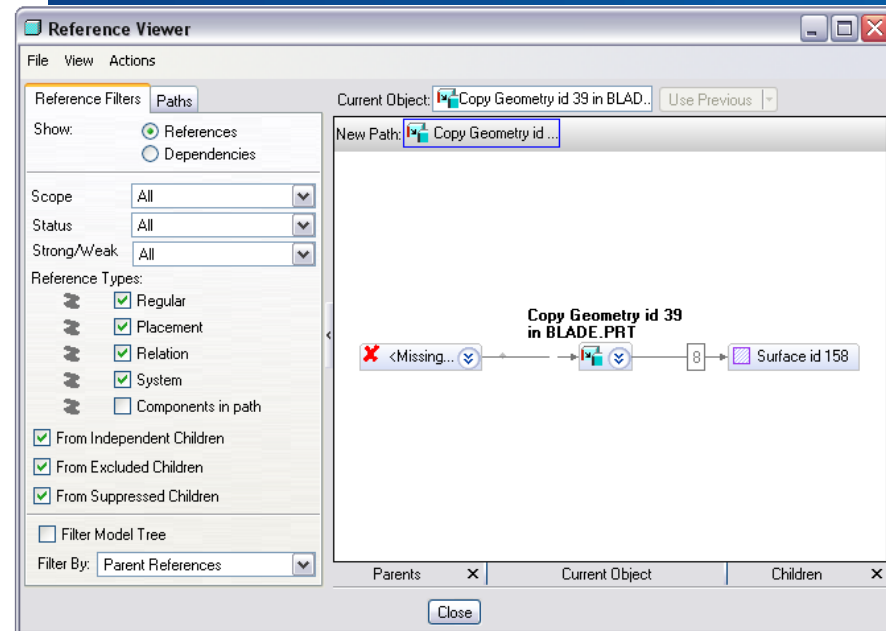
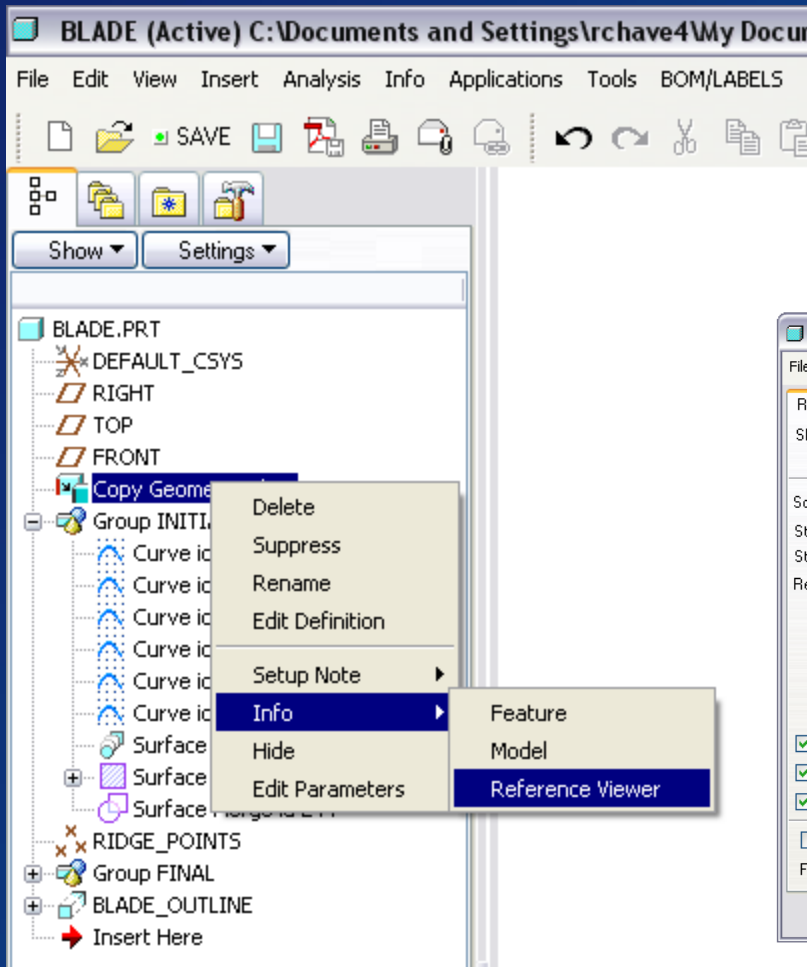
# ProE REFERENCE VIEWER (cont)

- The next step would be to identify any children features
- Select the Copy Geometry feature and open the Reference Viewer to identify its children
- Select the child feature to highlight in model



# ProE REFERENCE VIEWER (cont)

- Open Reference Viewer to identify any other External References still left and redefine as needed



# ProE REFERENCE VIEWER (cont)

- Typical missing dependencies
- Right-click on connecting line to bring up additional information
- “Break Dependency” may be available depending on type or reference. If used, item will no longer be considered a missing reference but may be “frozen” instead. Further data correction may be needed to correct frozen item.

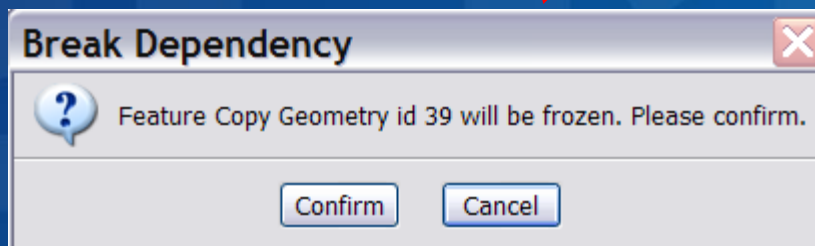
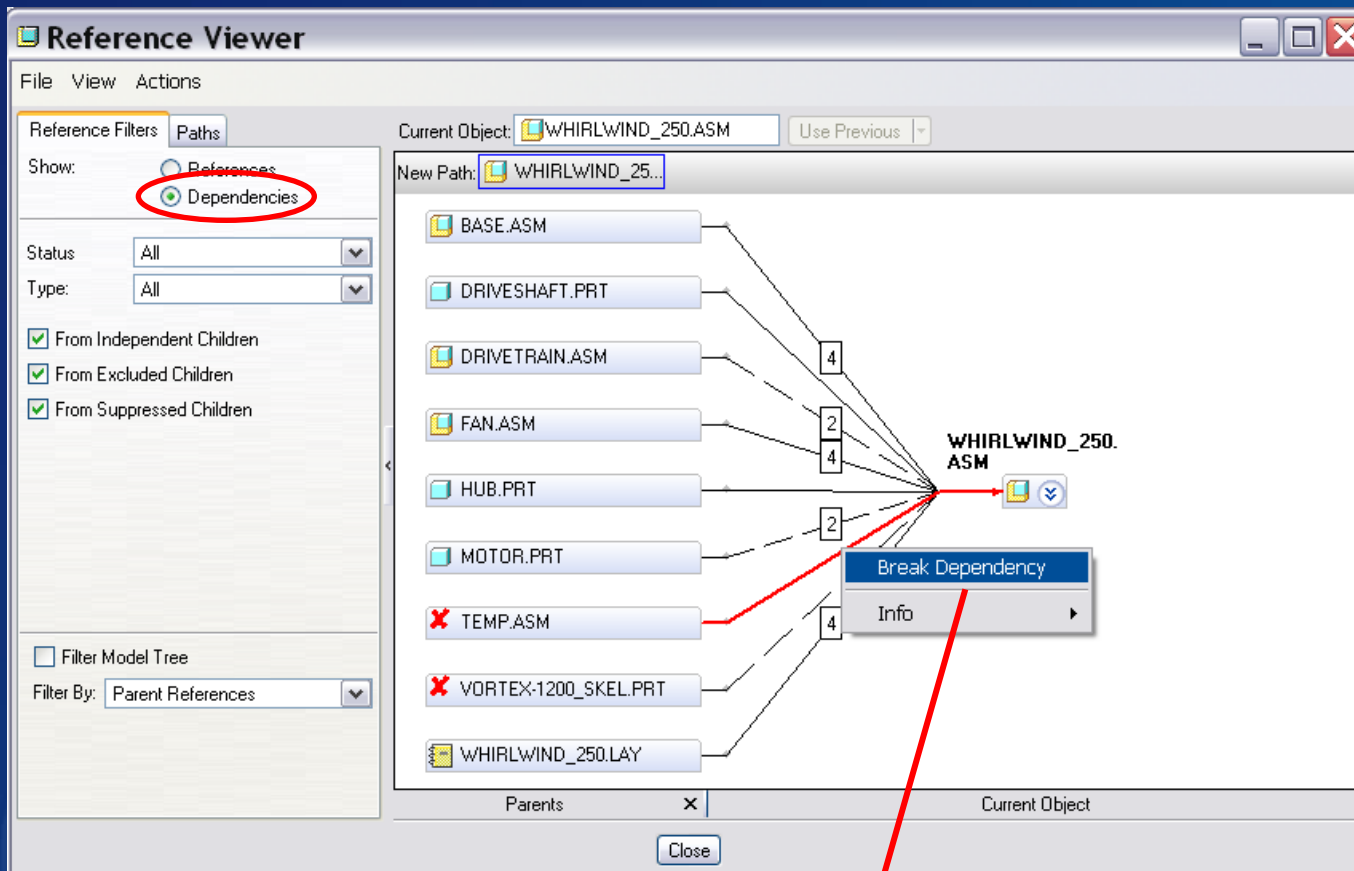
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## ProE REFERENCE VIEWER (cont)

- Dependencies between features and models are created by Pro/ENGINEER to enable work on Product Development Management systems.
- The following dependency types can be broken:
  - Restructure dependencies
  - Dependencies to unused substitute components
  - Dependencies to reference models of independent Data Sharing features
  - Dependencies to layouts
  - Dependencies to Interchanges assemblies
  - Dependencies of mirrored models to the original ones
  - Dependencies to old assembly cuts
  - Dependencies to replaced components

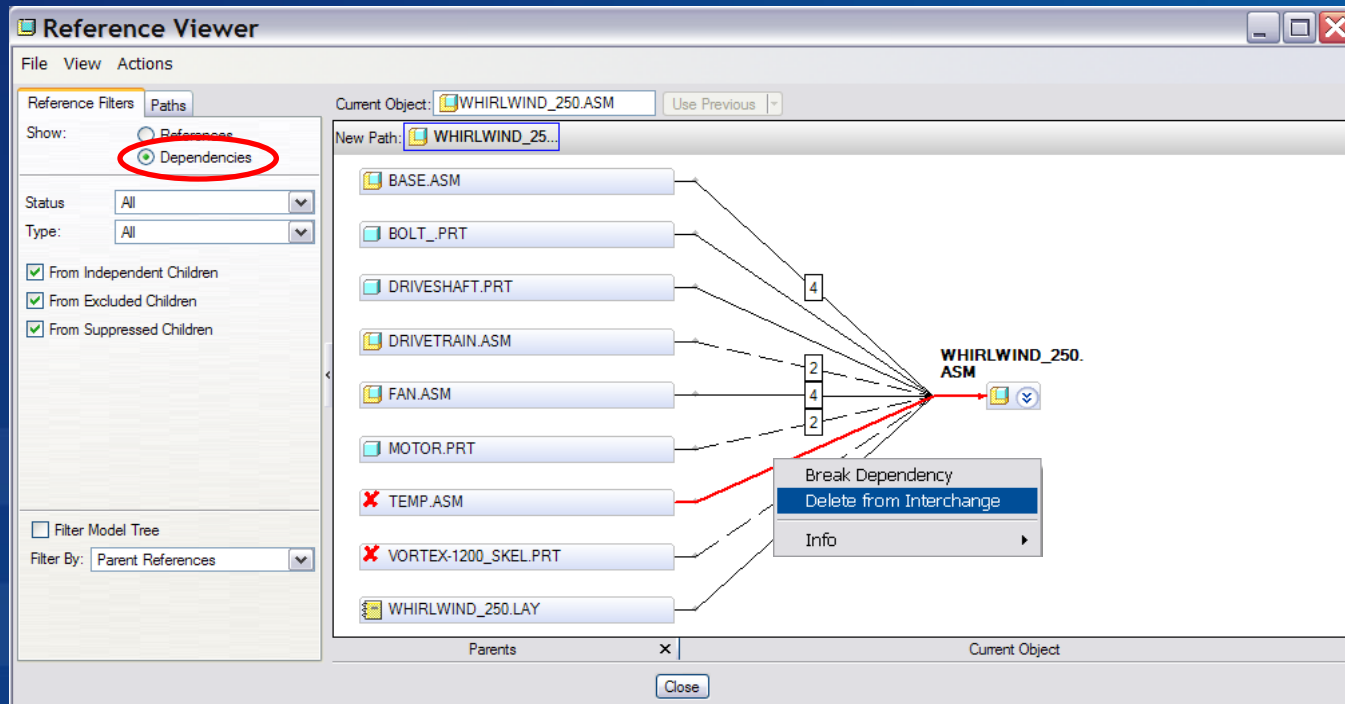


# ProE REFERENCE VIEWER (cont)



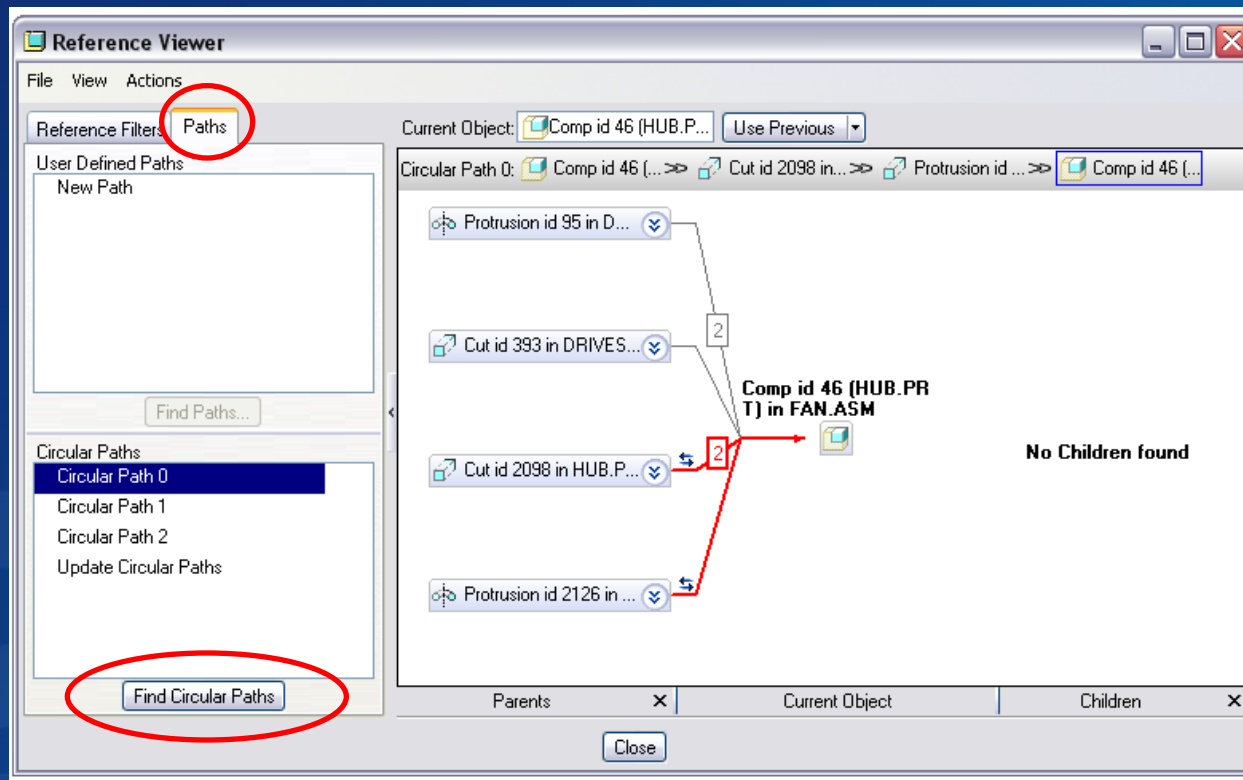
# ProE REFERENCE VIEWER (cont)

- Typical missing dependencies (Interchange type)
- Right-click on connecting line to bring up additional information
- “Delete from Interchange” may also be available
- Missing reference will then be removed from window



# ProE REFERENCE VIEWER (cont)

- Although Circular References will not generate Ghost Objects, the Reference Viewer may be very useful to identify and correct this error



# ProE REFERENCE VIEWER (cont)

- Reviewing details on Circular References

- Text style of info from .crc file created in working directory

Reference Viewer

File View Actions

Reference Filters: Paths

User Defined Paths

New Path

Find Paths...

Circular Paths

Circular Path 0

Circular Path 1

Circular Path 2

Update Circular Paths

Find Circular Paths

Current Object: Comp id 46 (HUB.P... Use Previous

Circular Path 0: Comp id 46 (...>> Cut id 2098 in...>> Protrusion id ...>> Comp id 46 (...

Protrusion id 95 in D...

Cut id 393 in DRIVES...

Comp id 46 (HUB.PRT) in FAN.ASM

Info

Display Full Path

No Children found

Protrusion id 2126 in ...

Parents < Current Object

Close

Full Path Display

FAN.ASM

Component id 46 (HUB.PRT)

HUB.PRT (Component id 46)

Cut id 2098

Surface id 2107

Displayed Full Path:

Component id 46 (HUB.PRT) -> Surface id 210

Close

→ \$ A circular reference between objects.

```
whirlwind_250.crc - Notepad
File Edit Format View Help
MODEL WHIRLWIND_250 :

3 CIRCULAR REFERENCE LOOPS FOUND

LOOP # 1
Depth of Loop is 4
Feature number 11 (COMPONENT, id = 46)
in model FAN refers to:
Feature number 37 (CUT, id = 2098)
in model HUB must follow after:
Feature number 5 (PROTRUSION, id = 46)
in model HUB refers to:
Feature number 11 (COMPONENT, id = 46)
in model FAN - circle closed.

LOOP # 2
Depth of Loop is 4
Feature number 11 (COMPONENT, id = 46)
in model FAN refers to:
Feature number 38 (PROTRUSION, id = 2126)
in model HUB must follow after:
Feature number 5 (PROTRUSION, id = 46)
in model HUB refers to:
Feature number 11 (COMPONENT, id = 46)
in model FAN - circle closed.


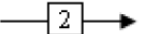




LOOP # 3
Depth of Loop is 4
Feature number 6 (PATTERN, id = 3008)
in model HUB must follow after:
Feature number 5 (PROTRUSION, id = 46)
in model HUB refers to:
Feature number 12 (PATTERN_1, id = 56)
in model FAN refers to:
Feature number 6 (PATTERN, id = 3008)
in model HUB - circle closed.
```

# ProE REFERENCE VIEWER (cont)

## •How to interpret Parent/Child relationships with Reference Viewer

### Reference Representations in the Parent/Child Graph

A reference is indicated by an arrow between two objects in the Parent/Child graph. When there is more than one reference, the number of references appears on the arrow.

Graphical Display	Definition
	A reference between objects.
	Multiple references between objects. The number of references is indicated in the box.
	An external reference.
	Colors can be assigned to distinguish between different reference types. By default, relation references are marked in red.
	A reference that is not updated. Make sure the parent object is in session and regenerate the assembly to update the reference.
	A circular reference between objects.

# MISSING DRAWING REFERENCES

- Even if no missing model references are found, drawing (.DRW) files can also have missing references.
- These missing drawing references will also appear in the PDM workspace as “Ghost” files
- No indications will be provided from within the model Reference Viewer
- Two possible solutions are available:
  - TPI 114763: Use of config.pro Parameters
  - TAN 137288: How to remove ghost object in Pro/ENGINEER drawing

# MISSING DRAWING REFERENCES (cont)

- TAN 137288 details how to remove as below:
  - Create a new drawing with the assembly or part model
    - Leave sheet 1 blank with just the template
  - >Insert > Shared Data > From File, select the "Problematic" drawing to be merged in the new drawing
    - This adds additional sheets starting with sheet 2
  - >Edit > Remove > Sheet
    - Enter sheet number 1 to remove the empty sheet 1.
  - Save the drawing and done.
- Any "Ghost" files still in the Workspace can now be removed as long as all missing relationships have been corrected

## MISSING DRAWING REFERENCES (cont)

- An additional method using config.pro settings on how to identify and remove drawing related ghost objects can be found on PTC Support Web Site (log-in required).
- TPI 114763: Use of config.pro Parameters "cleanup\_layout\_dependencies" and "cleanup\_drawing\_dependencies" to Remove Ghost Objects (References) From Layouts and Drawings in Pro/INTRALINK Workspace

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# MISSING DRAWING REFERENCES (cont)

- TPI 114763: Per PTC's Solution Details page
  - Using the config.pro options should only to be used in order to clean up dependencies which are known to be invalid. **DO NOT ENABLE THESE OPTIONS BY DEFAULT.**
  - Retrieval performance is significantly decreased when the cleanup options are enabled. Pro/ENGINEER will automatically search the Commonspace for each missing object during the retrieval. This is a secondary reason to not have these options set to Yes by default.

# WINDCHILL PDM PREFERENCE MANAGER

- Preferences that affect resolution of Incomplete Objects (during check-in of data) can be set with the Windchill Preference Manager
  - >Display > Incomplete object resolution
    - If set to "Ignore optional dependencies" (default), any CAD tool internal and non-required dependencies are ignored
    - If set to "Ignore optional reference dependencies," any reference dependencies are ignored.
    - If set to "Ignore internal dependencies only," only internal CAD tool dependencies are ignored.
    - If set to "Do not allow to ignore," no incomplete dependencies can be ignored.