Parametric Technology Corporation®

Creo™ Elements/Pro™ 5.0
J-Link Release Notes

August 2011

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About This Guide

This document provides release-specific information for J-Link for Creo Elements/Pro 5.0. It describes new methods in J-Link. It also includes critical changes that require investigation of existing applications and adjustment of existing code.

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Documentation for J-Link vi

Documentation for J-Link

J-Link documentation comes in the following forms:

- API Wizard in HTML format
- *J-Link User's Guide* in PDF format
- **Reference Documents** page at www.ptc.com/appserver/cs/doc/refdoc.jsp (Choose the product or document type)

The API Wizard and the *J-Link User's Guide* are on the Creo Elements/Pro product DVD-ROM. You can view the API Wizard with your browser. To view and print the PDF book, you must have Acrobat Reader installed.

PTC welcomes your suggestions and comments on its documentation. Send comments electronically to MCAD-documentation@ptc.com.

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Critical Technical Changes

This chapter describes the changes in Creo Elements/Pro 5.0 and J-Link that might require alteration of existing J-Link applications.

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pfcDetail.DetailSymbolInstItem.GetInstructions

The method **pfcDetail.DetailSymbolInstItem.GetInstructions** now takes a new *Boolean* argument `GiveParametersAsNames`. Set this argument to `true` to display symbolic representations of parameters and drawing properties in the symbol instance. Set it to `false` to display the actual text seen by the user. To ensure that the compilation succeeds, rebuild your existing J-Link applications calling the method **pfcDetail.DetailSymbolInstItem.GetInstructions**.

Printing Instructions

The interface `pfcModel.PlotInstructions` containing the instructions for plotting files has been deprecated. Existing J-Link methods for creating and accessing the instruction attributes have also been deprecated. Use the new interface type `pfcExport.PrinterInstructions` and its methods instead. Refer to the Superseded Methods section for the complete list of methods that have been deprecated.

The following new interface types have also been added:

- `pfcExport.PrintPrinterOption` for printer settings
- `pfcExport.PrintMdlOption` for the definition of the model for printing
- `pfcExport.PrintPlacementOption` for the placement options for use while printing
- `pfcExport.PrinterPCFOptions` for the definition of the printing options for a Plotter Configuration File (PCF)

No-Resolve Mode

Pro/ENGINEER Wildfire 5.0 introduces No-Resolve mode, wherein if a model and feature regeneration fails, failed features and children of failed features are created and the regeneration of other features continues. By default, Creo Elements/Pro operates in No-Resolve mode. However, J-Link does not support regeneration in this mode. If Creo Elements/Pro is running in No-Resolve mode, the methods **pfcSolid.Solid.Regenerate** and **pfcSolid.Solid.ExecuteFeatureOps** throw an exception `pfcExceptions.XToolkitBadContext`.

To continue with the behavior of Pro/ENGINEER Wildfire 4.0 in Resolve mode, set the configuration option `regen_failure_handling` to `resolve_mode` in the Creo Elements/Pro session. Setting the configuration option to switch to Resolve mode ensures the old behavior as long as you do not retrieve the models saved under No-Resolve mode. To consistently preserve the old behavior, use Resolve mode from the beginning and throughout your Creo Elements/Pro session.

Support for Model Display in Asynchronous Mode

In asynchronous mode, the following method is not supported: **pfcModel.ModelActionListener.OnBeforeModelDisplay**.

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New and Superseded Methods

This chapter describes new methods for J-Link for Creo Elements/Pro 5.0.

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New Methods

The following section describes the new J-Link methods.

2D Export

New Method	Description
pfcModel.pfcModel.MedusaExportInstructions_Create	Creates a new instructions object for export of a drawing in EXPORT_MEDUSA format (using **pfcModel.Model.Export**).
pfcModel.pfcModel.Export2DOption_Create pfcModel.Export2DOption.SetExportSheetOption pfcModel.Export2DOption.SetModelSpaceSheet pfcModel.Export2DOption.SetSheets	Accesses the options to export multiple sheets of a drawing to 2D formats.

3D Export

New Method	Description
pfcExport.pfcExport.CatiaPart3DExportInstructions_Create pfcExport.pfcExport.CatiaProduct3DExportInstructions_Create pfcExport.pfcExport.CatiaCGR3DExportInstructions_Create pfcExport.pfcExport.JT3DExportInstructions_Create pfcExport.pfcExport.ParaSolid3DExportInstructions_Create pfcExport.Export.UG3DExportInstructions_Create	Creates a new instructions object for import of the following 3D import formats: • EXPORT_CATIA_PART • EXPORT_CATIA_PRODUCT • EXPORT_CATIA_CGR • EXPORT_JT • EXPORT_PARASOLID • EXPORT_UG

Datum Features

New Method	Description
Datum Plane Feature	
pfcDatumPlaneFeat.DatumPlaneFeat.GetFlip pfcDatumPlaneFeat.DatumPlaneFeat.GetConstraints pfcDatumPlaneFeat.DatumPlaneConstraint.GetConstraintType pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneThroughConstraint_Create	Provides read access to the properties of the Datum Plane feature. Provides read access to the properties of the

New Method	Description
pfcDatumPlaneFeat.DatumPlaneThroughConstraint.GetThroughRef pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneNormalConstraint_Create pfcDatumPlaneFeat.DatumPlaneNormalConstraint.GetNormalRef pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneParallelConstraint_Create pfcDatumPlaneFeat.DatumPlaneParallelConstraint.GetParallelRef pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneTangentConstraint_Create pfcDatumPlaneFeat.DatumPlaneTangentConstraint.GetTangentRef pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneOffsetConstraint_Create pfcDatumPlaneFeat.DatumPlaneOffsetConstraint.GetOffsetRef pfcDatumPlaneFeat.DatumPlaneOffsetConstraint.GetOffsetValue pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneOffsetCoordSysConstraint_Create pfcDatumPlaneFeat.DatumPlaneOffsetCoordSysConstraint.GetCsysAxis pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneAngleConstraint_Create pfcDatumPlaneFeat.DatumPlaneAngleConstraint.GetAngleRef pfcDatumPlaneFeat.DatumPlaneAngleConstraint.GetAngleValue pfcDatumPlaneFeat.pfcDatumPlaneFeat.DatumPlaneSectionConstraint_Create pfcDatumPlaneFeat.DatumPlaneSectionConstraint.GetSectionRef pfcDatumPlaneFeat.DatumPlaneSectionConstraint.GetSectionIndex	Datum Plane feature. Provides read access to the properties of the Datum Plane feature.

New Method	Description
pfcDatumPlaneFeat.DatumPlaneDefaultXConstraint.DatumPlaneDefaultXConstraint_Create pfcDatumPlaneFeat.DatumPlaneDefaultYConstraint.DatumPlaneDefaultYConstraint_Create pfcDatumPlaneFeat.DatumPlaneDefaultZConstraint.DatumPlaneDefaultZConstraint_Create	
Datum Axis Feature	
pfcDatumAxisFeat.DatumAxisFeat.GetConstraints pfcDatumAxisFeat.pfcDatumAxisFeat.DatumAxisConstraint_Create pfcDatumAxisFeat.DatumAxisConstraint.GetConstraintType pfcDatumAxisFeat.DatumAxisConstraint.GetConstraintRef pfcDatumAxisFeat.DatumAxisFeat.GetDimConstraints pfcDatumAxisFeat.pfcDatumAxisFeat.DatumAxisDimensionConstraint_Create pfcDatumAxisFeat.DatumAxisDimensionConstraint.GetDimOffset pfcDatumAxisFeat.DatumAxisDimensionConstraint.GetDimRef	Provides read access to the properties of the Datum Axis feature.
General Datum Point Feature	
pfcDatumPointFeat.DatumPointFeat.GetFeatName pfcDatumPointFeat.DatumPointFeat.GetPoints pfcDatumPointFeat.GeneralDatumPoint.GetName pfcDatumPointFeat.pfcDatumPointFeat.DatumPointPlacementConstraint_Create pfcDatumPointFeat.GeneralDatumPoint.GetPlaceConstraints pfcDatumPointFeat.pfcDatumPointFeat.DatumPointDimensionConstraint_Create	Provides read access to the properties of the General Datum Point feature.
pfcDatumPointFeat.GeneralDatumPoint.GetDimConstraints pfcDatumPointFeat.DatumPointConstraint.GetConst	Provides read access to the properties of the General Datum Point feature.

New Method	Description
raintRef pfcDatumPointFeat.DatumPointConstraint.GetConstraintType pfcDatumPointFeat.DatumPointConstraint.GetValue	
Datum Coordinate System Feature	
pfcCoordSysFeat.CoordSysFeat.GetOriginConstraints pfcCoordSysFeat.pfcCoordSysFeat.DatumCsysOriginConstraint_Create pfcCoordSysFeat.DatumCsysOriginConstraint.GetOriginRef pfcCoordSysFeat.CoordSysFeat.GetDimensionConstraints pfcCoordSysFeat.pfcCoordSysFeat.DatumCsysDimensionConstraint_Create pfcCoordSysFeat.DatumCsysDimensionConstraint.GetDimRef pfcCoordSysFeat.DatumCsysDimensionConstraint.GetDimValue pfcCoordSysFeat.DatumCsysDimensionConstraint.GetDimConstraintType pfcCoordSysFeat.CoordSysFeat.GetOrientationConstraints pfcCoordSysFeat.pfcCoordSysFeat.DatumCsysOrientMoveConstraint_Create pfcCoordSysFeat.DatumCsysOrientMoveConstraint.GetOrientMoveConstraintType pfcCoordSysFeat.DatumCsysOrientMoveConstraint.GetOrientMoveValue pfcCoordSysFeat.CoordSysFeat.GetIsNormalToScreen pfcCoordSysFeat.CoordSysFeat.GetOffsetType pfcCoordSysFeat.CoordSysFeat.GetOnSurfaceType pfcCoordSysFeat.CoordSysFeat.GetOrientByMethod	Provides read access to the properties of the Datum Coordinate System feature.

Drawing Sheets

New Method	Description
pfcSheet.SheetOwner.GetSheetFormatDescr	Returns the model descriptor of the drawing format used for the specified drawing sheet.

Export to Faceted Format

New Method	Description
pfcExport.TriangulationInstructions.GetAngleControl pfcExport.TriangulationInstructions.SetAngleControl pfcExport.TriangulationInstructions.GetChordHeight pfcExport.TriangulationInstructions.SetChordHeight	Accesses and modifies the angle control and chord height for the exported facet drawings and models.

Export to PDF

New Method	Description
pfcExport.pfcExport.PDFExportInstructions_Create	Creates a new instructions object for export to PDF format (using **pfcModel.Model.Export**).
pfcExport.PDFExportInstructions.SetFilePath pfcExport.PDFExportInstructions.SetOptions	Accesses the instructions for export to PDF.
pfcExport.pfcExport.PDFOption_Create pfcExport.PDFOption.SetOptionType pfcExport.PDFOption.SetOptionValue	Accesses the options required for export to PDF.

Family Tables

New Method	Description
pfcFamily.FamilyMember.GetImmediateGenericInfo	Returns the model descriptor of the immediate generic model.
pfcFamily.FamilyMember.GetTopGenericInfo	Returns the model descriptor of the top generic model.
pfcFamily.FamilyMember.GetCellIsDefault	Determines if the value of the item in the specified cell is the default value. The default value is the value of the specified item in the generic model.

Model

New Method	Description
pfcSession.BaseSession.GetActiveModel	Returns the active Creo Elements/Pro model.

Model Items

New Method	Description
pfcSession.BaseSession.AllowDuplicateModelItems	Controls the creation of duplicate model items.

Post Regeneration Relations

New Method	Description
pfcModel.Model. GetPostRegenerationRelations	Lists the post-regeneration relations assigned to the model.

Printing Files

New Method	Description
Printing Instructions	
pfcExport.pfcExport.PrinterInstructions_Create	Creates the `pfcExport.PrinterInstructions` object.
pfcExport.PrinterInstructions.SetPrinterOption pfcExport.PrinterInstructions.SetPlacementOption pfcExport.PrinterInstructions.SetModelOption pfcExport.PrinterInstructions.SetWindowId	Accesses and modifies the plotting instructions.
Printer Options	
pfcExport.pfcExport.PrintPrinterOption_Create	Creates the `pfcExport.PrintPrinterOption` object.
pfcSession.BaseSession.GetPrintPrinterOptions	Returns the `pfcExport.PrintPrinterOption` object containing the printer options.
pfcExport.PrintPrinterOption.SetDeleteAfter pfcExport.PrintPrinterOption.SetFileName pfcExport.PrintPrinterOption.SetPaperSize pfcExport.Export.PrintSize_Create pfcExport.PrintSize.SetHeight pfcExport.PrintSize.SetWidth	Accesses and modifies the options for a specified printer.

New Method	Description
pfcExport.PrintSize.SetPaperSize pfcExport.PrintPrinterOption.SetPenTable pfcExport.PrintPrinterOption.SetPrintCommand pfcExport.PrintPrinterOption.SetPrinterType pfcExport.PrintPrinterOption.SetQuantity pfcExport.PrintPrinterOption.SetRollMedia pfcExport.PrintPrinterOption.SetRotatePlot pfcExport.PrintPrinterOption.SetSaveMethod pfcExport.PrintPrinterOption.SetSaveToFile pfcExport.PrintPrinterOption.SetSendToPrinter pfcExport.PrintPrinterOption.SetSlew pfcExport.PrintPrinterOption.SetSwHandshake pfcExport.PrintPrinterOption.SetUseTtf	
Placement Options	
pfcExport.pfcExport.PrintPlacementOption_Create	Creates the `pfcExport.PrintPlacementOption` object.
pfcSession.BaseSession.GetPrintPlacementOptions	Returns the `pfcExport.PrintPlacementOption` object containing the placement options.
pfcExport.PrintPlacementOption.SetBottomOffset pfcExport.PrintPlacementOption.SetClipPlot pfcExport.PrintPlacementOption.SetKeepPanzoom pfcExport.PrintPlacementOption.SetLabelHeight pfcExport.PrintPlacementOption.SetPlaceLabel pfcExport.PrintPlacementOption.SetScale pfcExport.PrintPlacementOption.SetShiftAllCorner pfcExport.PrintPlacementOption.SetSideOffset pfcExport.PrintPlacementOption.SetX1ClipPosition pfcExport.PrintPlacementOption.SetX2ClipPosition	Accesses and modifies the placement options. Accesses and modifies the placement options.

New Method	Description
pfcExport.PrintPlacementOption.SetY1ClipPosition pfcExport.PrintPlacementOption.SetY2ClipPosition	
Model Options	
pfcExport.pfcExport.PrintMdlOption_Create	Creates the `pfcExport.PrintMdlOption` object.
pfcSession.BaseSession.GetPrintMdlOptions	Returns the `pfcExport.PrintMdlOption` object containing the model options for printing purpose.
pfcExport.PrintMdlOption.SetDrawFormat pfcExport.PrintMdlOption.SetFirstPage pfcExport.PrintMdlOption.SetLastPage pfcExport.PrintMdlOption.SetLayerName pfcExport.PrintMdlOption.SetLayerOnly pfcExport.PrintMdlOption.SetMdl pfcExport.PrintMdlOption.SetQuality pfcExport.PrintMdlOption.SetSegmented pfcExport.PrintMdlOption.SetSheets pfcExport.PrintMdlOption.SetUseDrawingSize pfcExport.PrintMdlOption.SetUseSolidScale	Accesses and modifies the model options.
Plotter Configuration File (PCF) Option	
pfcExport.pfcExport.PrinterPCFOptions_Create	Creates the `pfcExport.PrinterPCFOptions` object.
pfcSession.BaseSession.GetPrintPCFOptions	Returns the `pfcExport.PrinterPCFOptions` object containing the printing options for a Plotter Configuration File (PCF).
pfcExport.PrinterPCFOptions.SetPrinterOption pfcExport.PrinterPCFOptions.SetPlacementOption pfcExport.PrinterPCFOptions.SetModelOption	Accesses and modifies the printing options for a Plotter Configuration File (PCF).

User Interface

New Method	Description
File > Open	

New Method	Description
pfcSession.BaseSession.UIRegisterFileOpen pfcUI.pfcUI.FileOpenRegisterOptions_Create pfcUI.FileOpenRegisterOptions.SetFileDescription pfcUI.FileOpenRegisterOptions.SetFileType pfcUI.FileOpenRegisterListener.FileOpenAccess pfcUI.FileOpenRegisterListener.OnFileOpenRegister	Adds a new file type in the Open dialog box.
File > Save	
pfcSession.BaseSession.UIRegisterFileSave pfcUI.pfcUI.FileSaveRegisterOptions_Create pfcUI.FileSaveRegisterOptions.SetFileDescription pfcUI.FileSaveRegisterOptions.SetFileType pfcUI.FileSaveRegisterListener.FileSaveAccess pfcUI.FileSaveRegisterListener.OnFileSaveRegister	Adds a new file type in the Save a Copy dialog box.
Navigation Area	
pfcSession.Session.NavigatorPaneBrowserAdd pfcSession.Session.NavigatorPaneBrowserIconSet pfcSession.Session.NavigatorPaneBrowserURLSet	Adds custom panes containing custom Web pages to the Navigation area.

Utility

New Method	Description
Pro/TOOLKIT DLL	
pfcSession.BaseSession.LoadProToolkitLegacyDll	Registers and starts a legacy Pro/TOOLKIT DLL that is not Unicode-compliant and built in the pre-Wildfire 4.0 environment.

Window Operations

New Method	Description
pfcSession.BaseSession.FlushCurrentWindow	Flushes the pending display commands on the current window.

Creo Elements/Pro Window

New Method	Description
Window ID	
pfcWindow.Window.GetId	Retrieves the ID of the Creo Elements/Pro window.

Superseded Methods

The following table lists superseded methods in this release.

Superseded Method	New Method
pfcModel.pfcModel.PlotInstructions_Create	pfcExport.pfcExport.PrinterInstructions_Create
pfcModel.PlotInstructions.SetPlotterName	pfcExport.PrinterInstructions.SetPrinterOption
pfcModel.PlotInstructions.SetOutputQuality	pfcExport.PrinterInstructions.SetPlacementOption
pfcModel.PlotInstructions.SetUserScale	pfcExport.PrinterInstructions.SetModelOption
pfcModel.PlotInstructions.SetPenSlew	pfcExport.PrinterInstructions.SetWindowId
pfcModel.PlotInstructions.SetPenVelocityX	
pfcModel.PlotInstructions.SetPenVelocityY	
pfcModel.PlotInstructions.SetSegmentedOutput	
pfcModel.PlotInstructions.SetLabelPlot	
pfcModel.PlotInstructions.SetSeparatePlotFiles	
pfcModel.PlotInstructions.SetPaperSize	
pfcModel.PlotInstructions.SetPageRangeChoice	
pfcModel.PlotInstructions.SetPaperSizeY	
pfcModel.PlotInstructions.SetFirstPage	
pfcModel.PlotInstructions.SetLastPage	

Obsolete Note Functions

Starting from Creo Elements/Pro 5.0 M100, the following Note functions are obsolete:

- pfcNote.NoteLeader
- pfcNote.NoteLeaders
- pfcNote.NotePlacement
- pfcNote.ParametricNotePlacement
- pfcNote.FreeNotePlacement
- Enumerated type `NoteLeaderType`

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Miscellaneous Technical Changes

The following changes in Creo Elements/Pro 5.0 can affect functional behavior in J-Link. PTC does not anticipate that these changes cause critical issues with existing J-Link applications.

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3D Import Formats

The class `pfcImport.NewModelImportType` now contains new 3D import formats. The J-Link method **pfcSession.BaseSession.ImportNewModel** supports the following new import formats:

- IMPORT_NEW_CATIA_PART
- IMPORT_NEW_UG
- IMPORT_NEW_PRODUCTVIEW
- IMPORT_NEW_CATIA_CGR
- IMPORT_NEW_JT

The class `pfcModel.IntfType` also contains new 3D feature import formats. The J-Link method **pfcSolid.Solid.CreateImportFeat** supports the following new import formats:

- INTF_ICEM
- INTF_ACIS
- INTF_DXF
- INTF_CDRS
- INTF_STL
- INTF_VRML
- INTF_PARASOLID
- INTF_AI
- INTF_CATIA_PART
- INTF_UG
- INTF_PRODUCTVIEW
- INTF_CATIA_CGR
- INTF_JT

Datum Features Properties

J-Link now provides read access to the properties of Datum features. The table below lists the Datum features supported and the new packages in which their properties and the corresponding read methods have been defined:

Datum Feature	J-Link Package
Datum Plane feature	`com.ptc.pfc.pfcDatumPlaneFeat`
Datum Axis feature	`com.ptc.pfc.pfcDatumAxisFeat`

Datum Feature	J-Link Package
Datum Point feature	com.ptc.pfc.pfcDatumPointFeat
Coordinate System feature	com.ptc.pfc.pfcCoordSysFeat

Deleting an Item from Simplified Representation

To delete an existing item in the simplified representation, set the action in the method **pfcSolid.SimpRep.SimpRepActionType** to NULL.

Export Formats

New export formats have been added to the class pfcModel.ExportType. The following table lists the new export formats and the new instructions object added for each format:

Export Format	Interface
EXPORT_MEDUSA	pfcExport.MedusaExportInstructions
EXPORT_CATIA_PART	pfcExport.CatiaPart3DExportInstructions
EXPORT_CATIA_PRODUCT	pfcExport.CatiaProduct3DExportInstructions
EXPORT_CATIA_CGR	pfcExport.CatiaCGR3DExportInstructions
EXPORT_JT	pfcExport.JT3DExportInstructions
EXPORT_PARASOLID	pfcExport.ParaSolid3DExportInstructions
EXPORT_UG	pfcExport.UG3DexportInstructions
EXPORT_PDF	pfcExport.PDFExportInstructions

Creo Elements/View Export Formats

The J-Link method **pfcModel.Model.Export** now supports export to any one of the Creo Elements/View formats listed in the next table. These formats have been defined in a new class pfcExport.ProductViewFormat.

Creo Elements/View Format	Type Constant
PVS	PV_FORMAT_PVS
ED	PV_FORMAT_ED
EDZ	PV_FORMAT_EDZ
PVZ	PV_FORMAT_PVZ

The attribute *PVExportOptions* has been added to the existing interface pfcExport.ProductViewExportInstructions. The values taken by this attribute are given by the new interface pfcExport.ProductViewExportOptions. This interface contains the attribute *PVFormat*, which can be set to any one of the Creo Elements/View formats listed above.

Export to PDF and U3D

J-Link now supports the export of Creo Elements/Pro drawings and solid models to PDF and U3D formats. A drawing can be exported as a 2D raster image embedded in a PDF file. The 3D models can be exported in the following ways:

- As a U3D model embedded in a one-page PDF file
- As 2D raster images representing saved views embedded in pages of a PDF file
- As a standalone U3D file

A new interface `pfcExport.PDFExportInstructions` containing all the instructions for export to PDF has been added. New PDF option types have also been defined in the class `pfcExport.PDFOptionType`.

Exporting to PDF with Save Options

The class **pfcExport.PDFSaveMode** specifies the PDF save options in the enumerated type `PDFOPT_PDF_SAVE` during the export of Creo Elements/Pro drawings and solid models to Portable Document Format (PDF). It takes the following values:

- `PDF_ARCHIVE_1`
- `PDF_FULL`

Layer Items

You cannot add the following items to a layer:

- `ITEM_SURFACE`
- `ITEM_EDGE`
- `ITEM_COORD_SYS`
- `ITEM_AXIS`
- `ITEM_SIMPREP`
- `ITEM_DTL_SYM_DEFINITION`
- `ITEM_DTL_OLE_OBJECT`
- `ITEM_EXPLODED_STATE`

Listing Groups in a Solid

The method **pfcSolid.Solid.ListGroups** returns the list of groups including UDFs in the solid.

New Types for pfcSession.BaseSession.ListFiles()

Starting with Pro/ENGINEER Wildfire 5.0 M040, the method **pfcSession.BaseSession.ListFiles()** has been enhanced to return instance objects when accessing Windchill workspaces or folders. A PDM location (for workspace or commonspace) must be passed as the directory path.

The following new types have been added under the enumerated type `FileListOpt`:

- FILE_LIST_ALL_INST—Same as the FILE_LIST_ALL option. It returns instances only for PDM locations.
- FILE_LIST_LATEST_INST—Same as the FILE_LIST_LATEST option. It returns instances only for PDM locations.

File List Options

Starting with Creo Elements/Pro 5.0 M100, the following values have been added to the enumerated type `FileListOpt`:

- FILE_LIST_ALL—Lists all the files.
- FILE_LIST_LATEST—Lists only the latest version of each file.

Obsolete Data Exchange Formats

The following data exchange formats and the corresponding type constants are no longer supported:

Data Exchange Format	Type Constant
CADAM	CADAM_CPTR_FILE CADAM_DIRECT_FILE CADAM_FILE IMPORT_2D_CADAM
PDGS	EXPORT_PDGS
CATIA	EXPORT_CATIA INTF_CATIA IMPORT_NEW_CATIA