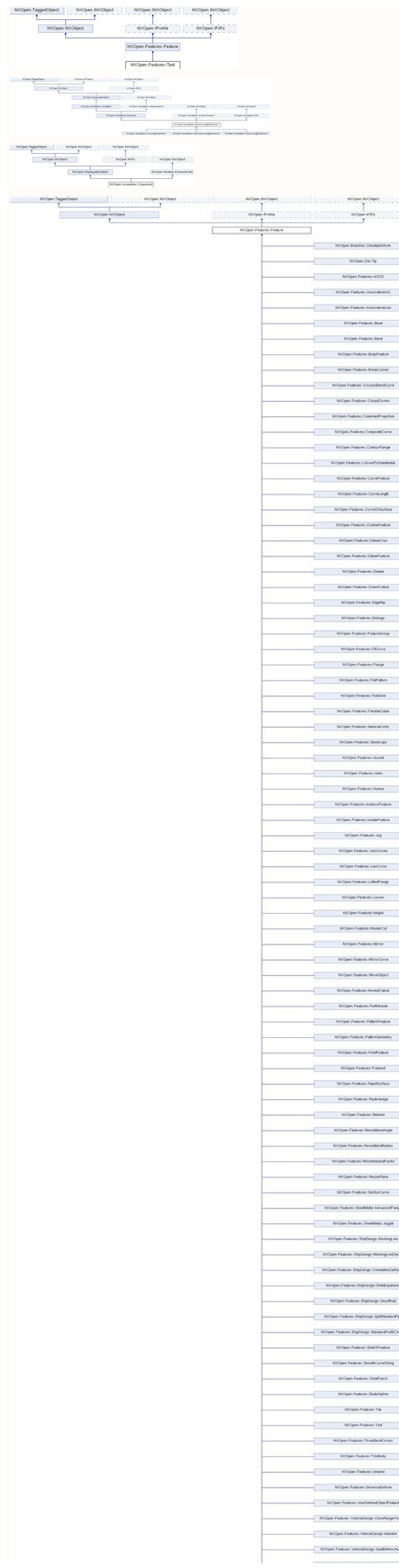
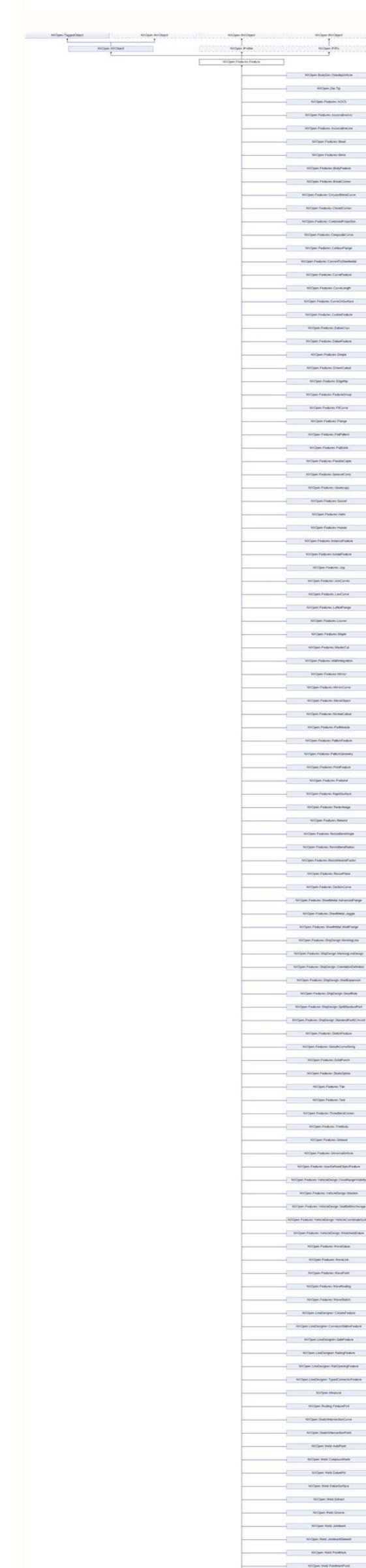


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NX Open is an API for Siemens' NX, which is computer-assisted design/engineering software. Use this tag for questions about using the NX Open API to get information or extend functionality to Siemens NX. Learn more... Top users Synonyms The automation tools and processes available in NX Drafting lend themselves to a great deal of user customization. A lot of basic customizations can be managed with customer defaults. More extensive customizations can be managed with a system of automation rules. There are two types of automation rules available: Simple - These rules do not require mathematical or complex logic and can be configured using customer defaults, for example, Annotation Distribution and Dimension rules. Knowledge Fusion (KF) - These rules require more complex logic, such as configuring drawing templates and booklets. Although they are constructed as a KF routine, they can be used to directly call a C, C++, or net program from the NX Open Application Programming Interface (API). For example, a KF rule can be used to assign the default orientation of a base view and the application of view scales and labels in a drawing template. The rules are saved with the template part and evaluated when the template is used to create drawings. For booklets, KF rules define the booklet names, booklet disciplines, types, and available templates. A repository of KF rules is found in the UGII\BASE\DI\draftingplus\automation\example directory. You can copy any of them to a writable location and customize them to your specifications. Applying rules to drawing templates You can imbed rules into custom templates to automate dimension, note, and symbol creation when the template is used to construct a drawing in another part. The KF rule applied to a template region affects the behavior of drafting object(s) associated with that region. A few of the examples you can utilize are: Template Region Type KF file View ViewScale.dfa Defines the scale (1:1) of the view added to the template region. ViewLabelText.dfa Defines the label content of a view. Table SampleReferenceTable.dfa Supplies table data to create and populate a tabular note when the template is used to create a drawing. Note NoteBuilderExample.dfa Adds text to the note region. Symbol SymbolDataExample.dfa Adds symbol data to a symbol region. Tip You can also apply a rule directly to an annotation or symbol that appears on the drawing by right-clicking the annotation or symbol and selecting Specify Rule. Only those rules available for that object type will be selectable. You can also manually create and position lines-based dimensions to drawing views with the Annotate Views and Distribute Annotations command. Booklet customization The Create Drawing Booklet wizard is built on a foundation of KF rules. These rules can be customized to tailor the options available in the wizard interface with your company-specific data. The rules are located in the UGII\BASE\DI\draftingplus\automation\dfa directory. The following table provides a list of commonly used KF files that can be modified to customize the options available in the Create Drawing Booklet wizard. KF file Description DrawingBooklet.dfa Controls the options in the Create Drawing Booklet wizard Setup, Template, Scope, and Attributes pages. You can do the following: Set the default value for the booklet name. Set the type of geometry that can be selected and added to a drawing. Set the template parts used by the wizard. Set part attributes that will be applied to each drawing part in the booklet. DrawingSetup.dfa Controls the options in the Create Drawing Booklet wizard Setup page. You can do the following: Set the default drawing standard for the drawings. Control the list of drawing type and drawing discipline displayed for the Drawing Style and Disciplines options. Note KF rules must be loaded into your NX session before launching the Create Drawing Booklet wizard. To do this, go to the directory that contains the rule files, change file type to All files and bring up the directory context menu. Select Knowledge Fusion Help for more information on creating and using KF rules. To access the latest configuration in the dfa file in your current NX session, start the Knowledge Fusion application, and then in an open area of the Knowledge Navigator, right-click and choose Reload All. Open UGS NX6 and check your local C++ journal code generation. Start recording journal and click on the actions you want to perform. Then is a cylinder creation in my example. Once you have completed stop recording journal and check code. In case of cylinder the generated code should look like this. Session->theSession->Session_GetSession(); Part->workPart->FacetedBodies()->DeleteTemporaryFacesAndEdges();Session->UndoMarkId markId1; markId1 = theSession->SetUndoMark(Session->MarkVisibilityVisible, "Start");Features::Feature *nullFeatures Feature(NULL); Features::CylinderBuilder *cylinderBuilder1; cylinderBuilder1 = workPart->Features()->CreateCylinderBuilder(); BooleanOption()->SetTargetBodies(targetBodies1);Session->SetUndoMark(markId1, markId1 = theSession->SetUndoMark(Session->MarkVisibilityVisible, "Start"));theSession->SetUndoMarkName(markId2, "Vector Dialog"); // Dialog_Begin Vector // Session->UndoMarkId markId3; markId3 = theSession->DeleteUndoMark(markId3, NULL);theSession->SetUndoMarkName(markId2, "Vector");theSession->DeleteUndoMark(markId2, NULL);Axis *axis1; axis1 = cylinderBuilder1->Axis();axis1->SetDirection(direction1);cylinderBuilder1->Diameter1->SetRightHandSide("50");cylinderBuilder1->Height1->SetRightHandSide("150");Session:>UndoMarkId markId4; markId4 = theSession->SetUndoMark(Session->MarkVisibilityVisible, "Cylinder");NXObject *nXObject1; nXObject1 = cylinderBuilder1->Commit();theSession->DeleteUndoMark(markId4, NULL);theSession->SetUndoMarkName(markId1, "Cylinder");cylinderBuilder1->Destroy();workPart->FacetedBodies()->DeleteTemporaryFacesAndEdges(); Now open Visual Studio, and create C++ DLL project with blank template. Create and add allmain.cpp file to your source files. And add the following functions to it. UGSDLLTEST API int usfrs { return (int)Session->LibraryUnloadOptionImmediately; } UGSDLLTEST API void usfrs(char * param,int * retcod,int param_len) { if(l!=0) MessageBox(NULL,"initialize error","L'init error",MB_OK|UF_DELETE); }where UGSDLLTEST EXPORTS #define UGSDLLTEST API _declspec(dllexport) #else #define UGSDLLTEST API _declspec(dllimport) #endif First function int usfrs ask unload() allows to choose between different resource deallocation type. You can choose between LibraryUnloadOptionAtTermination, LibraryUnloadOptionExplicitly and LibraryUnloadOptionImmediately. If you don't specify this function LibraryUnloadOptionAtTermination is set by default. This means UGS does not release exe or dll file until you close it. If you specify LibraryUnloadOptionImmediately UGS immediately releases after use dll/exe file you've been working through NX Open invocation. This is how we want in NX Open application development. Second function void usfrs(char * param,int * retcod,int param_len) is specific NX Open function which you have to provide to have custom NX open application working in your NX environment. It doesn't matter whether you're creating dll or exe, this function is a start point for any NX Open unmanaged application. See more for NX Open documentation for more details. Add libfun.lib, libopenprt.lib, libxopenccp.lib to linker properties so that application could be linked. VS linker properties Now add this code to your usfrs function and test it. A 150 mm height cylinder should appear. To do this click from menu file->execute->NxOpen, and ensure that a DLL file filter is set in the application type. If you don't see execute menu, change your UGS type to advanced with full menus Additional improvements Play around with code generated by journal and find unnecessary lines. Result uglsimart (Aeronautics) (OP) 7 Mar 21 17:47 Hello All.. 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SmartObject::UpdateOptionWithinModeling);theSession->DeleteUndoMark(markId3, NULL);theSession->SetUndoMarkName(markId2, "Vector");theSession->DeleteUndoMark(markId2, NULL);Axis *axis1; axis1 = cylinderBuilder1->Axis();axis1->SetDirection(direction1);cylinderBuilder1->Diameter1->SetRightHandSide("50");cylinderBuilder1->Height1->SetRightHandSide("150");Session:>UndoMarkId markId4; markId4 = theSession->SetUndoMark(Session->MarkVisibilityVisible, "Cylinder");NXObject *nXObject1; nXObject1 = cylinderBuilder1->Commit();theSession->DeleteUndoMark(markId4, NULL);theSession->SetUndoMarkName(markId1, "Cylinder");cylinderBuilder1->Destroy();workPart->FacetedBodies()->DeleteTemporaryFacesAndEdges(); Now open Visual Studio, and create C++ DLL project with blank template. Create and add allmain.cpp file to your source files. And add the following functions to it. UGSDLLTEST API int usfrs { return (int)Session->LibraryUnloadOptionImmediately; } UGSDLLTEST API void usfrs(char * param,int * retcod,int param_len) { if(l!=0) MessageBox(NULL,"initialize error","L'init error",MB_OK|UF_DELETE); }where UGSDLLTEST EXPORTS #define UGSDLLTEST API _declspec(dllexport) #else #define UGSDLLTEST API _declspec(dllimport) #endif First function int usfrs ask unload() allows to choose between different resource deallocation type. You can choose between LibraryUnloadOptionAtTermination, LibraryUnloadOptionExplicitly and LibraryUnloadOptionImmediately. If you don't specify this function LibraryUnloadOptionAtTermination is set by default. This means UGS does not release exe or dll file until you close it. If you specify LibraryUnloadOptionImmediately UGS immediately releases after use dll/exe file you've been working through NX Open invocation. This is how we want in NX Open application development. Second function void usfrs(char * param,int * retcod,int param_len) is specific NX Open function which you have to provide to have custom NX open application working in your NX environment. It doesn't matter whether you're creating dll or exe, this function is a start point for any NX Open unmanaged application. See more for NX Open documentation for more details. Add libfun.lib, libopenprt.lib, libxopenccp.lib to linker properties so that application could be linked. VS linker properties Now add this code to your usfrs function and test it. A 150 mm height cylinder should appear. To do this click from menu file->execute->NxOpen, and ensure that a DLL file filter is set in the application type. If you don't see execute menu, change your UGS type to advanced with full menus Additional improvements Play around with code generated by journal and find unnecessary lines. Result uglsimart (Aeronautics) (OP) 7 Mar 21 17:47 Hello All.. Start using NX1953 and Visual Studio 2019 Community edition. To re-compile old UGOpen C++ files , first I need to install NXOPEN wizards The way I've been doing was : - copy ...UGOPEN\libs to "C:\Program Files (x86)\Microsoft Visual Studio" This case, when I try to create a new project , I can not see the UGOpen templates in the available template list. I've been using VS Professional or VS Enterprise by this time .. It's my first attempt to use Community edition.. I wonder if I should continue with VS 2019 Pro instead .. Need suggestions on this.. Thanks in advance .. Regards .. Thank you for helping keep Eng-Tips Forums free from inappropriate posts.The Eng-Tips staff will check this out and take appropriate action. Page 2 Low-Volume Rapid Injection Molding With 3D Printed Molds Learn methods and guidelines for using stereolithography (SLA) 3D printed molds in the injection molding process to lower costs and lead time. Discover how this hybrid manufacturing process enables on-demand mold fabrication to quickly produce small batches of thermoplastic parts. Download Now Design for Additive Manufacturing (DFAM) Examining how the principles of DFAM upend many of the long-standing rules around manufacturability - allowing engineers and designers to place a partâ€™s function at the center of their design considerations. Download Now Industry Perspective: Education and Metal 3D Printing Metal 3D printing has rapidly emerged as a key technology in modern design and manufacturing, so itâ€™s critical educational institutions include it in their curricula to avoid leaving students at a disadvantage as they enter the workforce. Download Now Taking Control of Engineering Documents This ebook covers tips for creating and managing workflows, security best practices and protection of intellectual property, Cloud vs. on-premise software solutions, CAD file management, compliance, and more. Download Now

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