

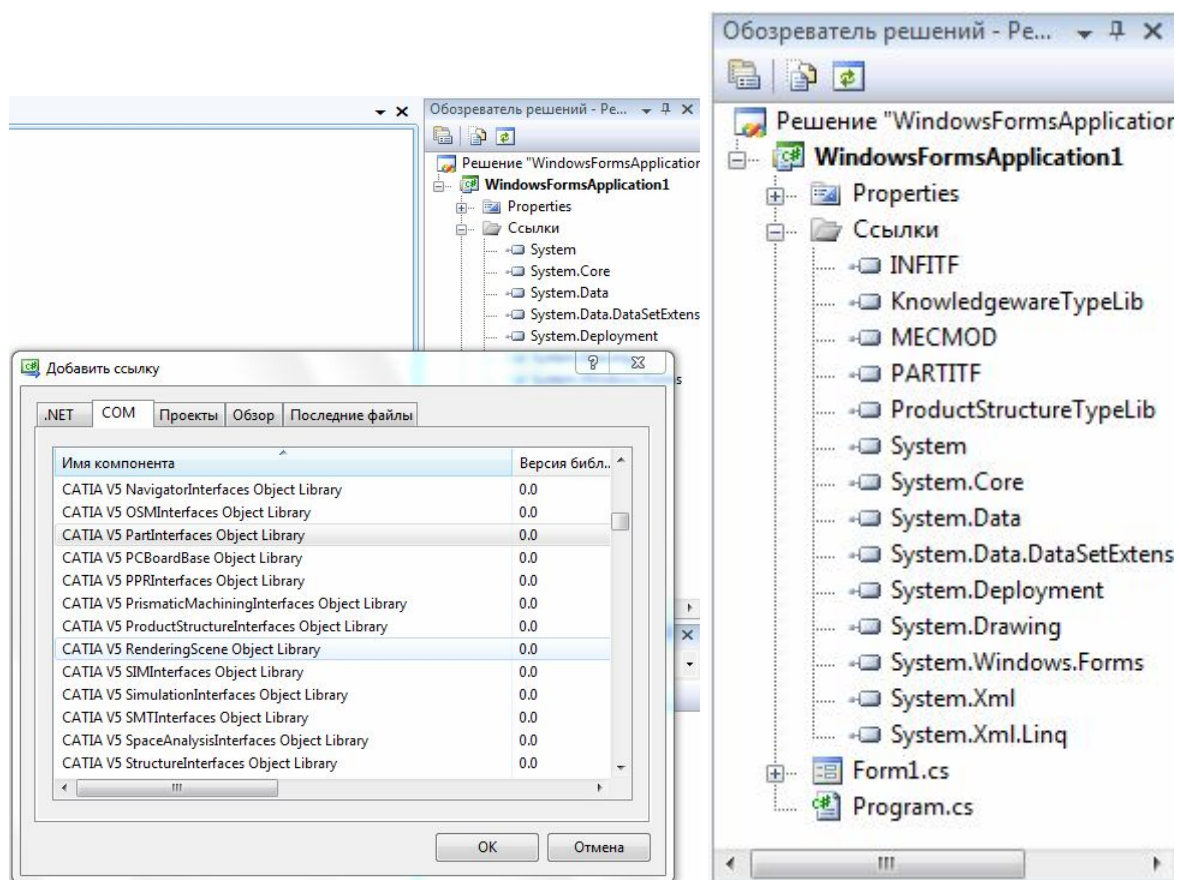
Windows application development on C# using CATIA V5 DLLs

In this article I made simple review of ways how to add references of CATIA COM DLLs to a new C# project. A way depends on how CATIA was installed on PC. Actually, if CATIA was installed using setup files all COM DLLs should be represented on COM tab of add reference dialog window (Paragraph I). If CATIA was moved from another PC like portable application you have to extract DLLs using tlbimp.exe of Microsoft SDK tools (Paragraph II).

I. CATIA V5 is installed on PC

1. Start new project in C#.

2. Add needed references to your project: you just need to add “CATIA V5 PartInterfaces Object Library” and/or “CATIA V5 DraftingInterfaces Object Library” on COM tab, other DLLs will be added automatically.



3. Now you are ready to code using CATIA V5 object model. Refer to V5Automation.chm to know base information. After compiling your code, DLLs you added as references will copy to result EXE folder of your project's structure.

II. CATIA V5 isn't installed, but used like portable application

1. First you need to prepare DLLs you will use in your project as references. It depends on CATIA branches you are going to interact. I usually interact with PartDesign and Drafting branches. Let's try to get needed DLLs.

- a) make sure there are PartTypeLib.tlb and DraftingTypeLib.tlb libraries in
\\intel_a\\code\\bin\\

b) to extract PartTypeLib.dll create following bat file:

```
set TLBIMP="C:\Program Files\Microsoft SDKs\Windows\v6.0A\Bin\tlbimp.exe"
set CATIA=C:\Program Files\Dassault Systemes\B19\intel_a\code\bin
set OUT=C:\Temp
set LIBNAME=PartTypeLib
%TLBIMP% "%CATIA%\%LIBNAME%.tlb" /out:%OUT%\%LIBNAME%.dll
```

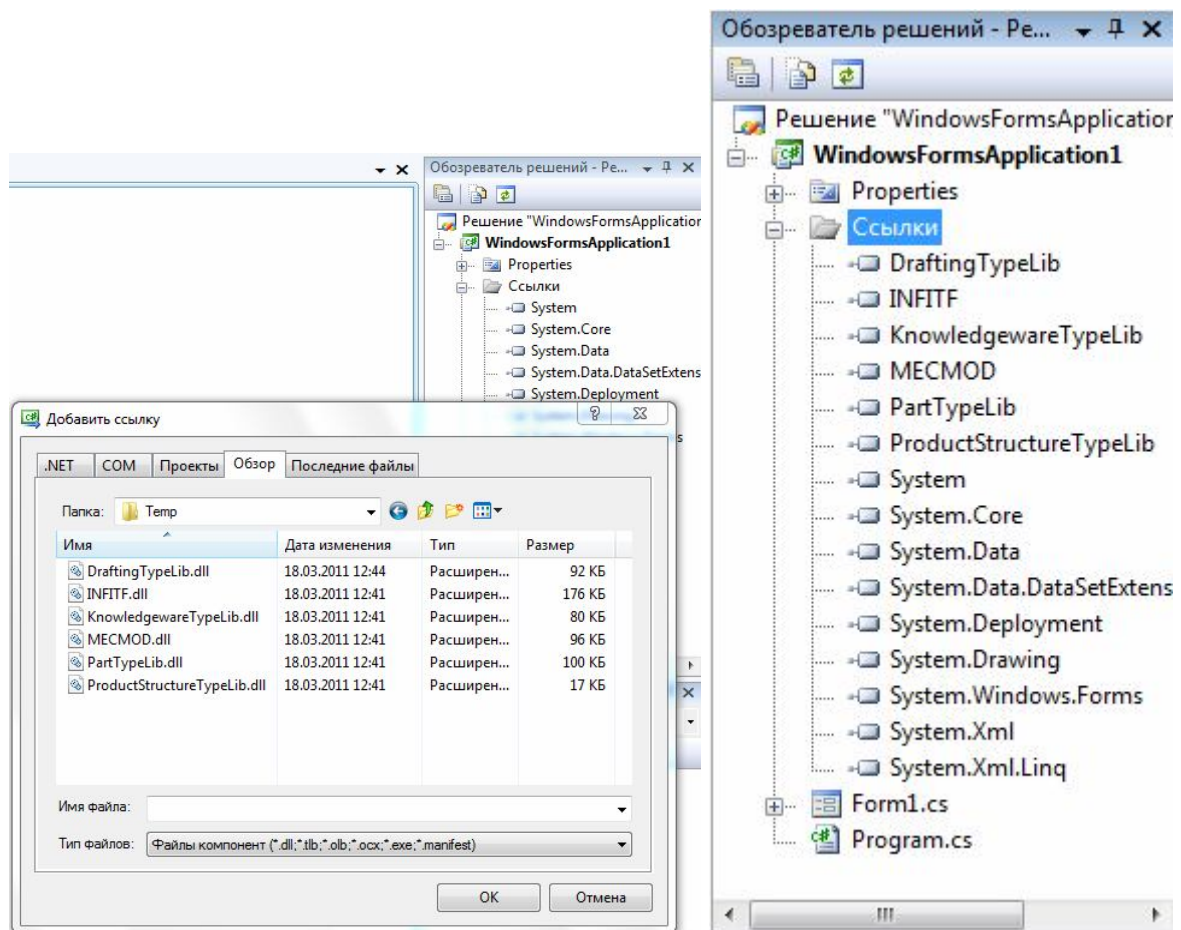
c) run this bat file to get PartTypeLib.dll in C:\Temp; you will see some more DLLs
PartTypeLib.dll depends on

d) edit fourth line of the bat file: **set LIBNAME=DraftingTypeLib** to extract DraftingTypeLib.dll

e) now there are files in C:\Temp: PartTypeLib.dll, DraftingTypeLib.dll, MECMOD.dll, ProductStructureTypeLib.dll, KnowledgewareTypeLib.dll, INFITF.dll

2. Start new project in C#

3. Add DLLs from C:\Temp as references to your project:



4. If you didn't have CATIA V5 installed before, you have to register CATIA as OLE-server in your Windows OS. Run "cnext.exe /regserver" using Command prompt.

5. Now you are ready to code using CATIA V5 object model. Refer to V5Automation.chm to know base information. After compiling your code, DLLs you added as references will copy to result EXE folder of your project's structure.

For example, let's add new body to opened CATPart file.
Add using clause:

```
using PartTypeLib;
using INFITF;
using MECMOD;
using DraftingTypeLib;
using System.Runtime.InteropServices;
```

Add a button on a form and fill Button1_Click method:

```
// ** Connect to CATIA V5
INFITF.Application catiaApp;
try
{
    catiaApp = (INFITF.Application)Marshal.GetActiveObject("CATIA.Application");
}
catch
{
    MessageBox.Show("CATIA V5 isn't launched");
    return;
}
// ^^

// ** Connect to opened PartDocument
PartDocument part_doc = null;
try
{
    part_doc = (PartDocument)catiaApp.ActiveDocument;
}
catch
{
    MessageBox.Show("Active document isn't a PartDocument");
    return;
}
// ^^

Part part = part_doc.Part;
part.Bodies.Add();
part.Update();
```