



# **PDM Workbench**

## **PDM Workbench Release 19.0.0 for Aras Innovator**

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### **Release Notes**

Version 1

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# Preface

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## About this Document

This document provides general release information for the PDM Workbench.

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## Related Documents

The following manuals contain information about installation, administration, usage, and customization of the PDM Workbench:

Manual Title	Version
<i>PDM Workbench Installation &amp; Administration Manual</i>	19.0
<i>PDM Workbench User Manual</i>	19.0

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## Your Comments are Welcome

Please feel free to tell us your opinion; we are always interested in improving our publications. Mail your comments to:

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# CHAPTER 1

## Platform Support

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### Supported Platforms

CATIA V5 Version V5-6R2022, V5-6R2023, V5-6R2024, and V5-6R2025

(other revisions on request):

CATIA V5 Client V5-6R2022	Windows 10 (64Bit), Windows 11 (64Bit)
CATIA V5 Client V5-6R2023	Windows 10 (64Bit), Windows 11 (64Bit)
CATIA V5 Client V5-6R2024	Windows 10 (64Bit), Windows 11 (64Bit)
CATIA V5 Client V5-6R2025	Windows 10 (64Bit), Windows 11 (64Bit)

Important notice:

CATIA V5-6R2014 SP2 has been retracted by Dassault Systèmes and is not supported. Please use SP3 instead.

CATIA V5-6R2022. Minimum service pack: SP1

Server Installation of Aras Innovator:

(Other service packs on request)

Aras Innovator Server 12 SP09	Windows Server 2019, Windows Server 2016, Windows Server 2012
Aras Innovator Server 22 to 34	Windows Server 2022, Windows Server 2019, Windows Server 2016

T-Systems licman21 license manager:

T-Systems licman21	Please refer to the Licman documentation.
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# CHAPTER 2

## Functional Changes

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### Construction Space

The Construction Space feature enables the user to set a construction space (assembly/installation space) to filter CAD structures.

The construction space itself represents an axis-aligned bounding-box which is defined by a min point and a max point. This axis-aligned bounding box represents a virtual three-dimensional cuboid, whose sides are the boundaries for the construction space.

Only structures which are located inside the construction space or intersect the construction space are loaded into the CAD application. Because of that the Construction Space feature is integrated into the “Open in CATIA” functionality. The feature is compatible with both CAD and BOM mode for “Open in CATIA”.

### Configuration

When installing the “Open in CATIA” data model the Construction Space feature is already included. Configurations might need to be done at the “PwbConstructionSpace” life cycle. *Please be aware that names of the life cycle states should not be modified to ensure the functionality of the Construction Space feature.* The identities to promote life cycle states can be modified to your needs.

### Access Rights

In the TOC, the Construction Space can be found at **Administration → PDM Workbench → Construction Space**.

The identity “All Employees” has access to the Construction Space ItemType. Restrictions can be made via the TOC editor.

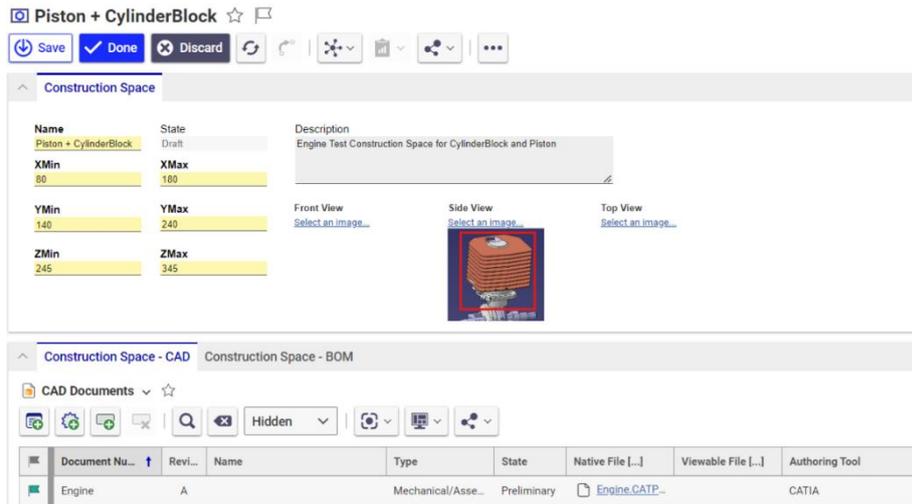
The usage of the Construction Space ItemType is recommended for the usage of the Construction Space feature.

Construction Space items which are in the “Released” state can be accessed by all users. In the “Draft” state only the creator can use the Construction Space for testing. If only creators should be allowed to access the Construction Space, then it should be left in the “Draft” state. All users of the “Aras PLM” identity can promote the life cycle state of the construction space. In the “Obsolete” state the construction space is no longer shown. The identities to promote life cycle states can be modified in the life cycle map.

### Usage

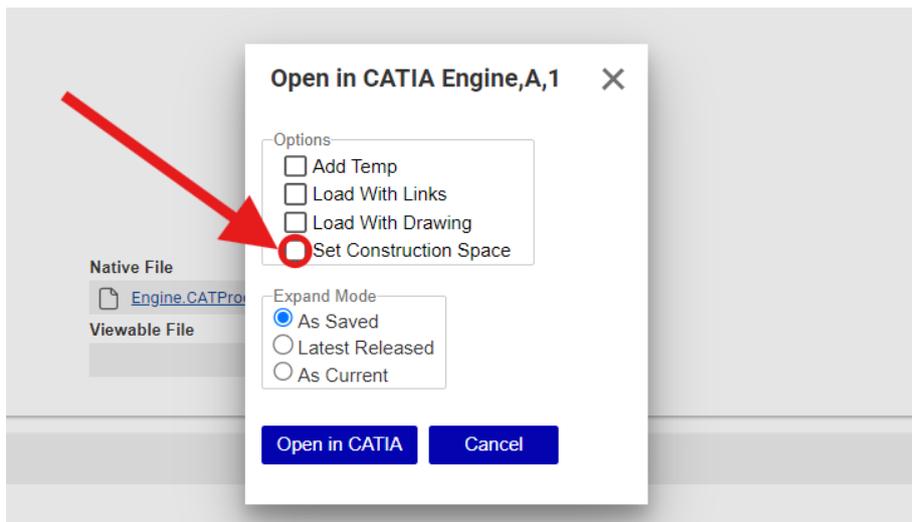
In *Picture 1: Create a “Construction Space” item*, the fields marked yellow (Name and Bounding Box) are mandatory. Optionally a description and thumbnails of three different views can be set.

In the Relationship Tab the user can link CAD Documents and Parts to the construction space item. The construction space can then be found when using the “Open In CATIA” functionality with the linked CAD Documents or Parts.



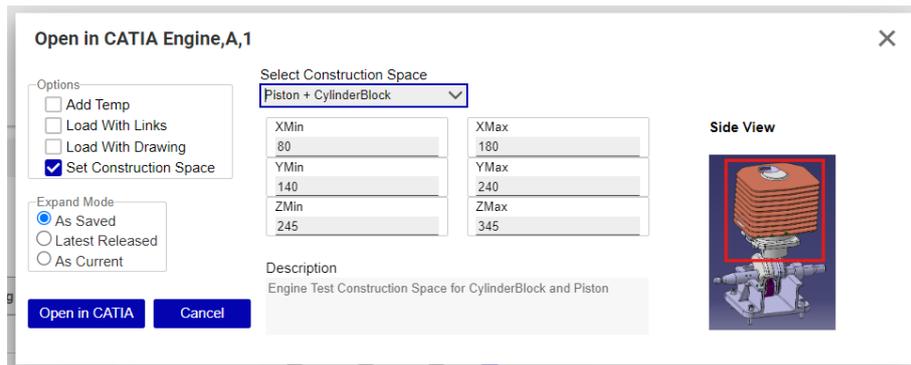
**Picture 1: Create a “Construction Space” item**

In *Picture 2: “Open In CATIA” option selection* you can see the “Set Construction Space” option. To use the Construction Space feature you have to select “Set Construction Space”.



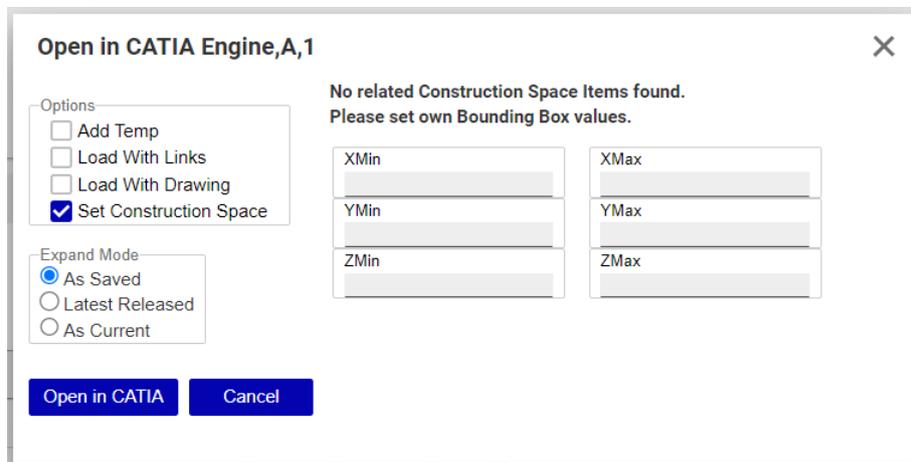
**Picture 2: “Open In CATIA” option selection**

The previously created construction space linked to the CAD file from which the “Open in CATIA” function was opened is shown when the construction space is selected in the dropdown list. Please be aware that the bounding box values in *Picture 3: “Open in CATIA” with predefined Construction Space* are editable but not saved in the Construction Space item. Permanent changes should be made in the construction space item.



**Picture 3: “Open in CATIA” with predefined Construction Space**

The message “No related Construction Space items found. Please set own Bounding Box values.” appears if there is no Construction Space item linked to the current file. In this case the user can manually set a bounding box by filling the bounding box values into the input mask (see *Picture 4: “Open In CATIA” without predefined Construction Space*).



**Picture 4: “Open In CATIA” without predefined Construction Space**

With a click on “Open in CATIA” the structures within or cutting the construction space are loaded into the CAD application.

## Compatibility with ACI

With ACI it is possible to export CATIA V5 data from 3DEXPERIENCE and import the data into Aras Innovator or to export CATIA V5 data from Aras Innovator and import the data into 3DEXPERIENCE. The imported data from 3DEXPERIENCE can be used with PDM Workbench. Data generated with PDM Workbench can be used in 3DEXPERIENCE.

If you want to continue working with both systems, you should always change a certain geometry in the same system and only use this geometry unchanged in the other system. This is because converting data always reduces the quality of the data.

To distinguish between original CATIA V5 data and CATIA V5 data from 3DEXPERIENCE PDM Workbench uses the `external_owner` property.

Data created by PDM Workbench uses  
`external_owner = T-Systems.Mechanical.CATIA`

## Data created by ACI uses

external\_owner = T-Systems.Mechanical.3DExperience

By default a query in PDM Workbench will only find objects with external\_owner=T-Systems.mechanical.CATIA. To find other objects, you must extend the *dataSource* for *ExternalOwner* in your PWB Schema.xml file:

```
<dataSource name="ExternalOwner" type="ValueList">
  <value
    name="T-Systems.Mechanical.CATIA,T-Systems.Mechanical.3DExperience"
    displayName=""/>
</dataSource>
```

With this change PDM Workbench query will also find objects with external\_owner=T-Systems.mechanical.3DExperience.

By default PDM Workbench loads all CATParts / CATProducts / CATDrawings with a *external\_owner* other than *T-Systems.mechanical.CATIA* in read only mode.

If the default behavior needs to be changed the default implementation can be overwritten by defining a C# server method whose name corresponds to the value of the server PWB setting "CustomMethod\_GetCustomItemInfo":

CustomMethod_GetCustomItemInfo	PwbCus_GetCustomItemInfo
--------------------------------	--------------------------

**Picture 5: Sample CustomMethod\_GetCustomItemInfo configuration**

## Sample method:

```
using (var PwbServerApiObj = new PwbServerAddin.PwbServerApi(this))
{
    PwbServerApiObj.Log("Called method 'PwbCus_GetCustomItemInfo' -> ");

    string PdmActionStr = getProperty("PdmAction");
    PwbServerApiObj.Log(" -> PdmActionStr:'" + PdmActionStr + "'");

    string InputObjsStr = getProperty("CustomItemInfoInputObjs");
    PwbServerApiObj.Log(" -> InputObjsStr:'" + InputObjsStr + "'");

    IDictionary<string, IDictionary<string, string>> ConfigDict =
        new Dictionary<string, IDictionary<string, string>>();

    var ItemList = PwbServerApiObj.QueryItemsForCustomItemInfo(InputObjsStr);
    foreach (var ItemObj in ItemList)
    {
        PwbServerApiObj.Log(" -> '"+ItemObj.getType()+"' / '" + ItemObj.getID() + "'");

        IDictionary<string, string> CurrentItemConfigDict;
        GetItemConfigSettings(PwbServerApiObj, ItemObj, out CurrentItemConfigDict);

        ConfigDict.Add(ItemObj.getID(), CurrentItemConfigDict);
    }

    IDictionary<string, string> OutputInfoDict = new Dictionary<string, string>();

    OutputInfoDict.Add(
        "CustomItemInfoObjConfigSettings",
        PwbServerApiObj.StringDictDictToString(ConfigDict));

    return PwbServerApiObj.DialogAttrsDictionaryToItem(OutputInfoDict);
}

private void GetItemConfigSettings(
    PwbServerAddin.PwbServerApi PwbServerApiObj,
    Item ItemObj,
    out IDictionary<string, string> ItemConfigDict)
{
    ItemConfigDict = new Dictionary<string, string>();

    // By default, the same calls as in the hardcoded server method.
    int LockStatus = ItemObj.getLockStatus();

    bool IsEditAllowed = (LockStatus == 1);
    bool IsClaimAllowed = (LockStatus == 0);
    bool ClaimAsNewGeneration = false;

    // Check for standard parts and for template files
    if ((PwbServerApiObj.ItemIsStandardPart(ItemObj)) &&
        (!PwbServerApiObj.IsUserStandardPartAdmin()))
```

```

    {
        IsEditAllowed = false;
        IsClaimAllowed = false;
    }

    if ((PwbServerApiObj.ItemIsTemplateFile(ItemObj)) &&
        (!PwbServerApiObj.IsUserTemplateFileAdmin()))
    {
        IsEditAllowed = false;
        IsClaimAllowed = false;
    }

    // Released items
    bool IsReleased = false;
    if ((ItemObj.getProperty("is_released") == "1" ) &&
        String.IsNullOrEmpty(ItemObj.getProperty("source_id"))) // not a relation
    {
        IsEditAllowed = false;
        IsClaimAllowed = true;
        ClaimAsNewGeneration = true;
        IsReleased = true;
    }

    // Superseded items should also not be modified.
    if (PwbServerApiObj.ItemIsSuperseded(ItemObj))
    {
        IsEditAllowed = false;
        IsClaimAllowed = false;
    }

    // Reconnected items must not be modified
    if (PwbServerApiObj.IsInReconnectContext())
    {
        IsEditAllowed = false;
        IsClaimAllowed = false;
    }

    // handle additional external owners like T-Systems.Mechanical.3DExperience
    if (String.IsNullOrEmpty(ItemObj.getProperty("source_id")) && // not a relation
        "Document" != ItemObj.getType() && // not a designTable
        PwbServerApiObj.GetExternalOwnerFromCadClient() !=
        PwbServerApiObj.GetExternalOwner(ItemObj)) // wrong external_owner
    {
        IsEditAllowed = false;
        IsClaimAllowed = false;
        ClaimAsNewGeneration = false;
    }

    if (IsEditAllowed || IsClaimAllowed)
    {
        PwbServerApiObj.CheckForEnvironmentAttributes(
            ItemObj,
            ref IsEditAllowed,
            ref IsClaimAllowed,
            ref ClaimAsNewGeneration);
    }

    // Debug info
    ItemConfigDict.Add("ItemNumber", ItemObj.getProperty("item_number", ""));
    ItemConfigDict.Add("Type", ItemObj.getType());
    ItemConfigDict.Add("Id", ItemObj.getID());

    bool UpdateAllowed = !IsReleased;

    if (!UpdateAllowed)
    {
        ClaimAsNewGeneration = true;
    }

    bool NewGenHasAlreadyBeenCreated =
        (ItemObj.getProperty("pwb_claimed_is_new_gen")=="1");
    bool CreateNewGenAtUpdate = ((!UpdateAllowed) || (!NewGenHasAlreadyBeenCreated));

    // Returning the values.
    ItemConfigDict.Add("IsEditAllowed", IsEditAllowed.ToString().ToLower());
    ItemConfigDict.Add("IsClaimAllowed", IsClaimAllowed.ToString().ToLower());

    ItemConfigDict.Add("ClaimAsNewGeneration",
        ClaimAsNewGeneration.ToString().ToLower());
    ItemConfigDict.Add("CreateNewGenAtUpdate",
        CreateNewGenAtUpdate.ToString().ToLower());
}

```

---

## Retrieval and Mapping of Inertia Attributes for non-BOM in BOM CATParts

PWB configuration settings `UseBomPartStructure=true` and  
`UseNonBom3DCadDocs=true`:

The inertia data are now also read from non-BOM in BOM CATParts

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# CHAPTER 3

## Data Model Changes

The following data model changes are introduced in this release of the PDM Workbench.

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### Changed Forms

“PwbOpenInCATIA\_BOM\_DialogForm” and “PwbOpenInCATIA\_CAD\_DialogForm”, added optional support to receive and select Construction Space items or to define a Bounding Box.

---

### Added Forms

“PwbConstructionSpace”, as an input form to create Construction Space items and to link Construction Space items to Parts or CAD Documents.

---

### Added ItemType

“PwbConstructionSpace”, as item representing the Bounding Box of the Construction Space. Thumbnails and a description can be set. The name of the Construction Space is the identifier.

---

### Added LifeCycle

“PwbConstructionSpace”, to set Access Rights for the Construction Space.

---

### Added RelationshipTypes

“PwbConstructionSpaceBOM” and “PwbConstructionSpaceCAD” to link the related files to Construction Space items.

---

### Changed Client Methods

“PwbOpenInCATIA\_Part” and “PwbOpenInCATIA\_CAD”, to support the Construction Space feature.

---

## Added Client Methods

“PwbOnChangeConstructionSpace”, triggered when a Construction Space is selected from the dropdown list.

“PwbOnClickSetConstructionSpace”, triggered when “Set Construction Space” is selected from the options menu.

## Changed Server Method

“PwbGetStructureAml”, to support the Construction Space feature.

## Added Server Methods

“PwbGetNeighborhoodAml”, as equivalent to the existing OpenInCAD method “PwbGetStructureAml”. The method is required to get an AML response from the PwbServer containing structures which are located within or intersect the Construction Space.

“PwbGetCSValues”, to get the values of the selected Construction Space item.

“PwbGetRelatesCSNames”, to get Construction Space item-names, which are linked to the selected Part or CAD.

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# CHAPTER 4

## Bug Fixes

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Version 19.0.0