

IM472654: Inventor Customer Success Program 项目介绍及分享

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讲师介绍

欧特克（中国）软件研发有限公司

刘静宜，多年制造业行业工作经验，2007年加入欧特克，任职华东区销售部门技术工程师，2008年转入市场发展部，担任制造业产品技术顾问；2012年加入Inventor产品研发部门，目前担任Inventor产品经理一职，参与Inventor产品的发展方向规划和负责部分Inventor产品的研发项目。

课程内容及目的

课程内容

Inventor Customer Success 项目介绍;
与产品团队互动
Inventor Customer Success项目的典型成果分享

目的

了解与Inventor产品的Customer Success 项目;
了解Inventor团队与用户的沟通渠道;
有效沟通的方法建议

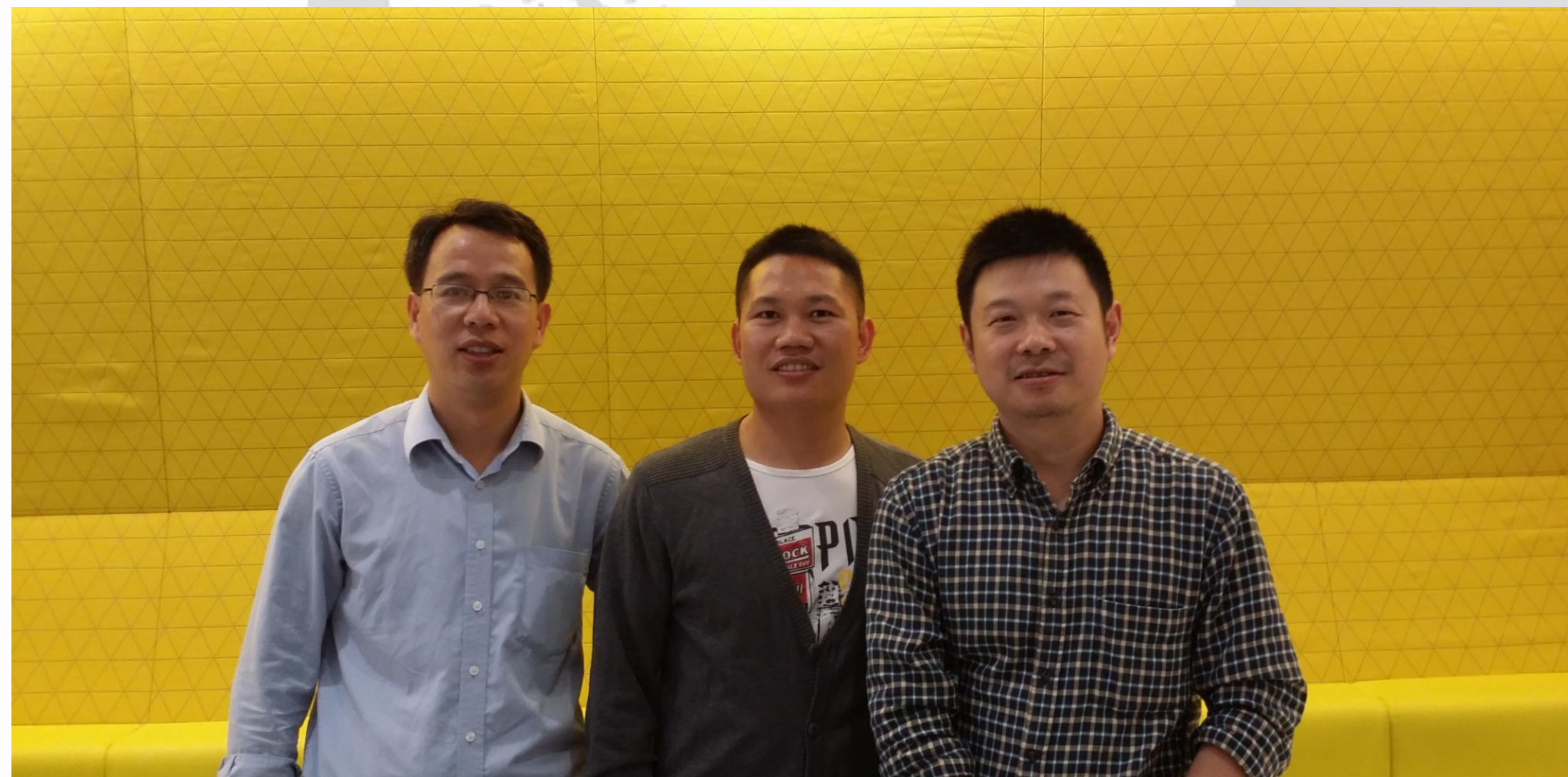
Inventor Customer Success 项目

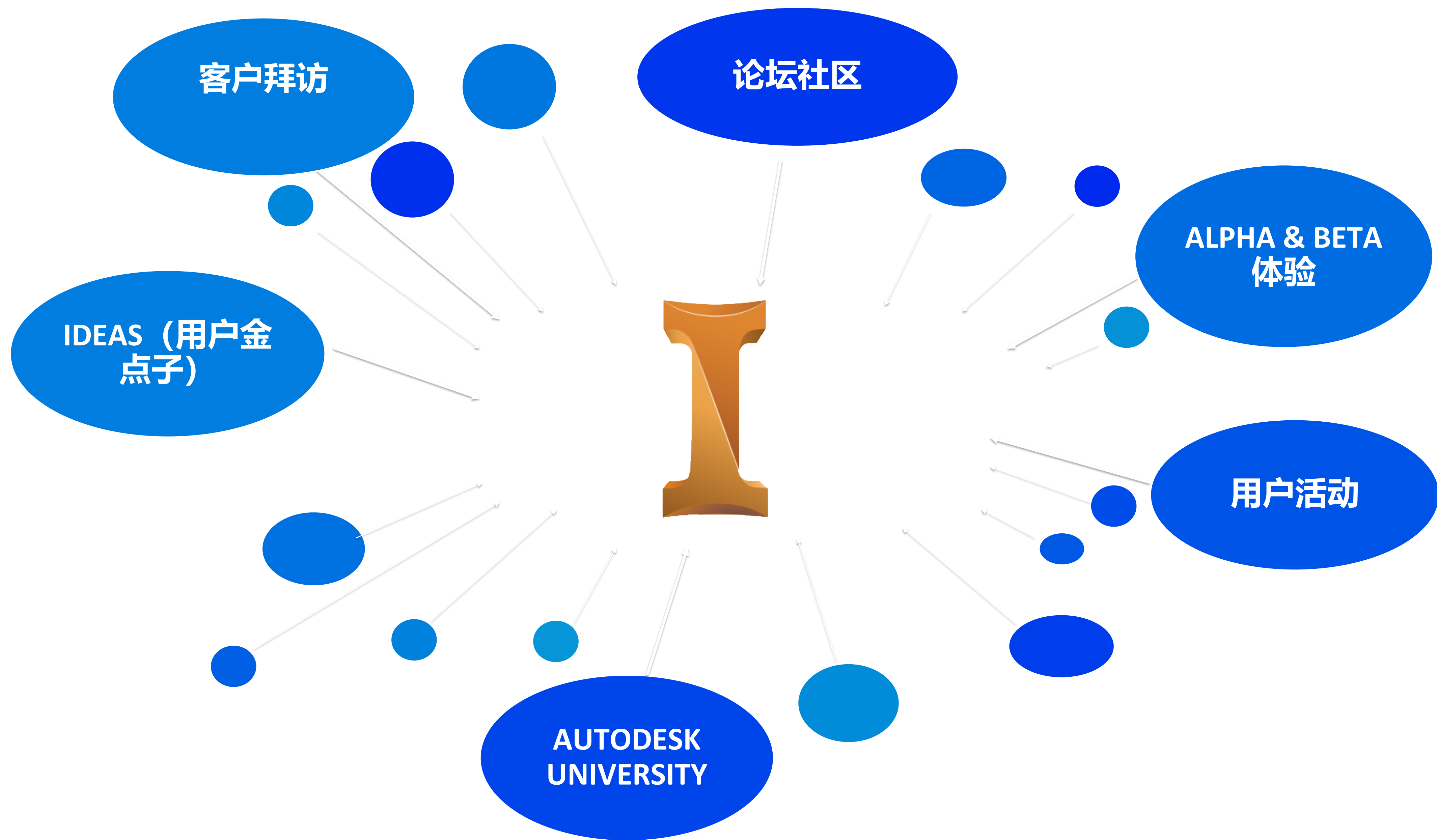


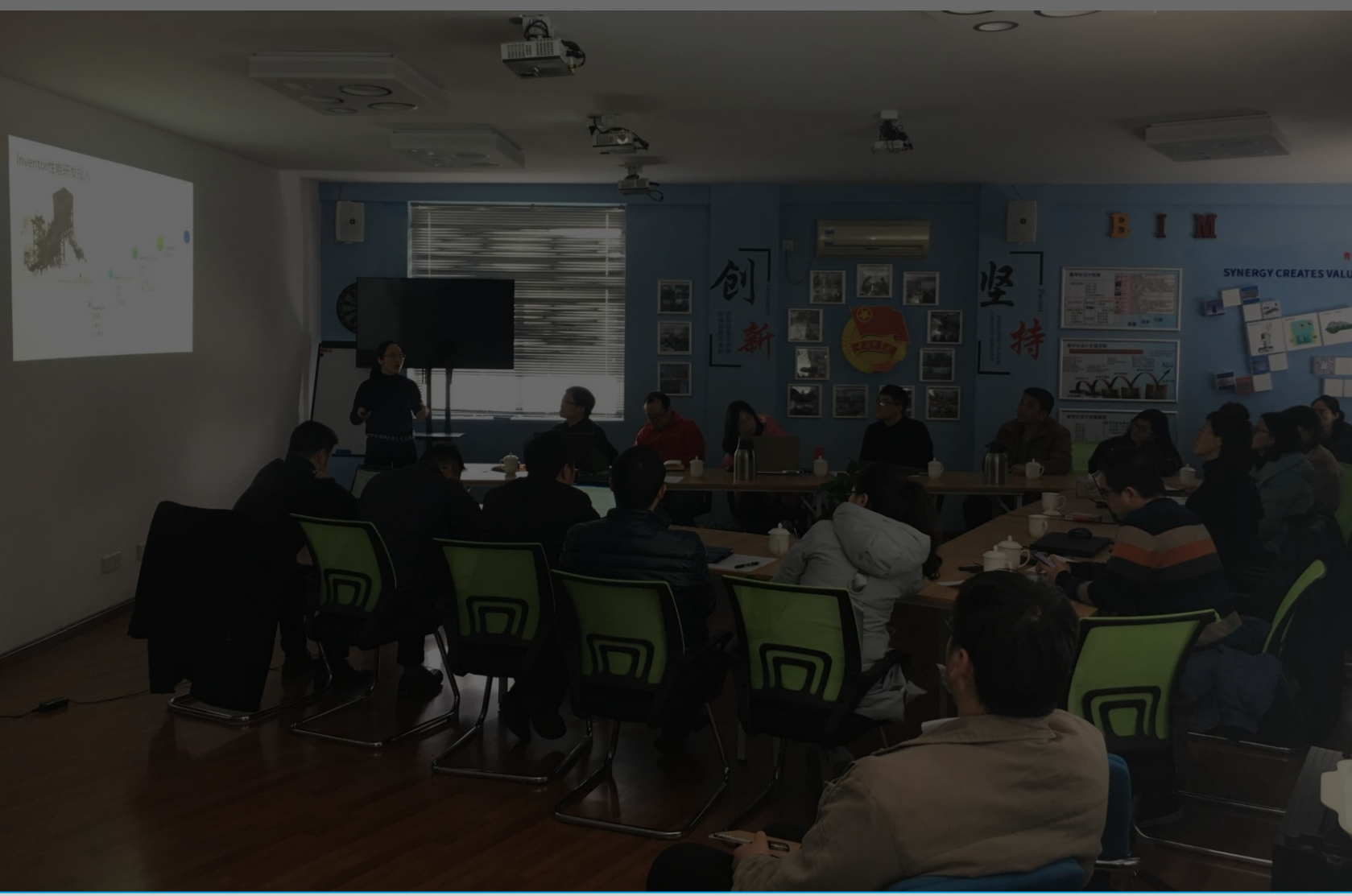
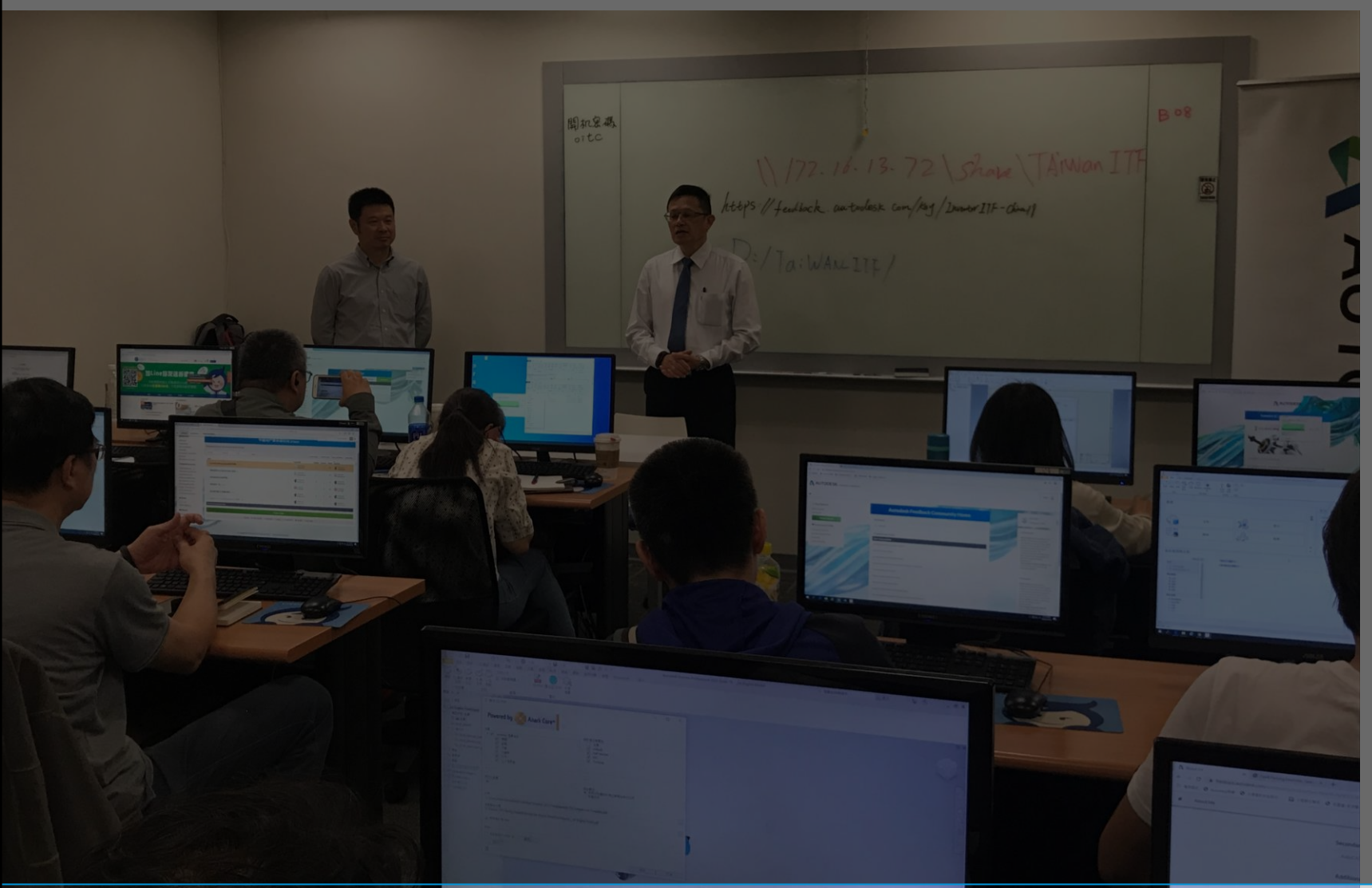
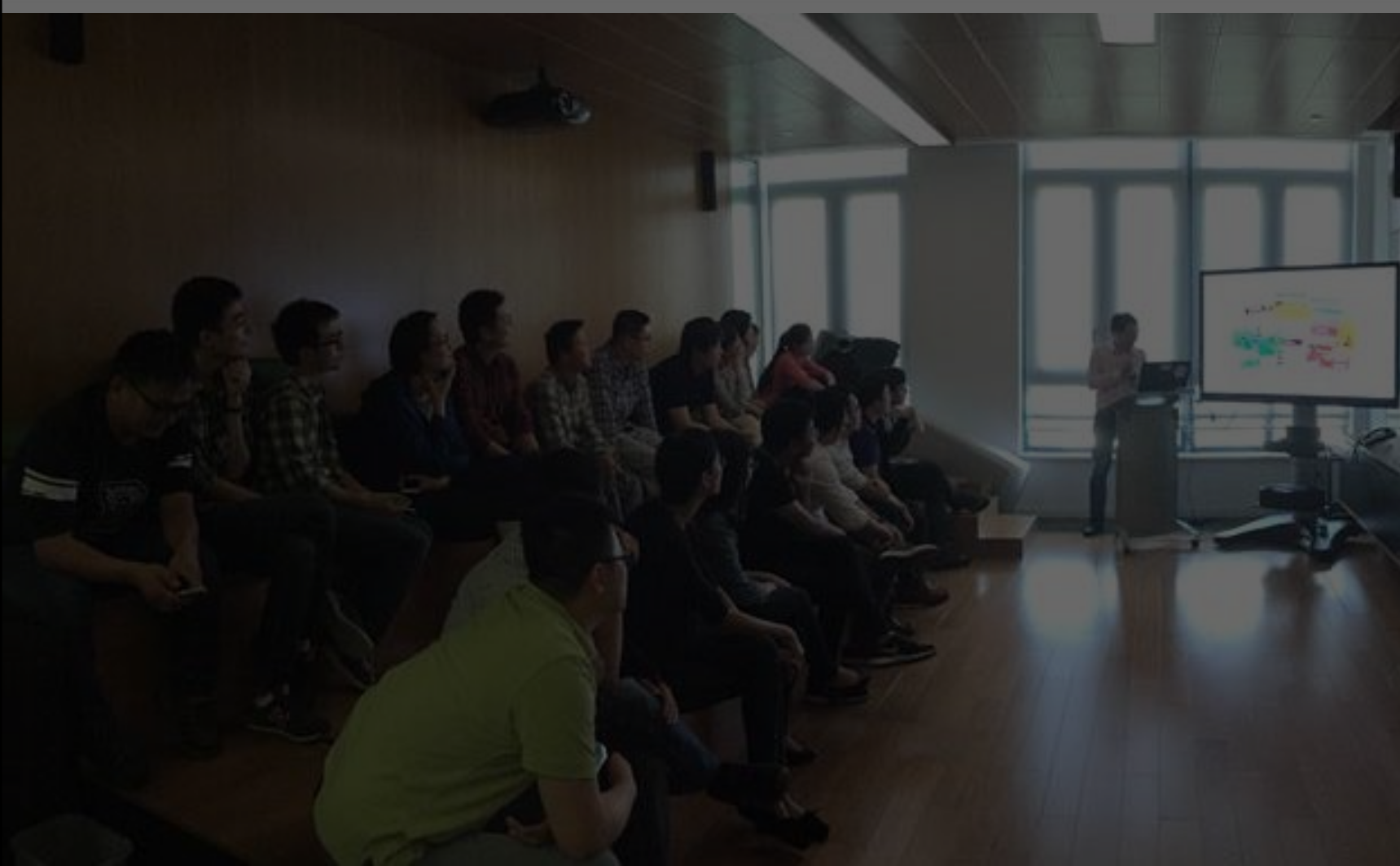
Inventor Customer Success项目

- 用户成功才能产品成功;
- 成立开发项目组, 解决存在已久的用户需求;★
- 通过用户互动, 加强用户的设计验证;★

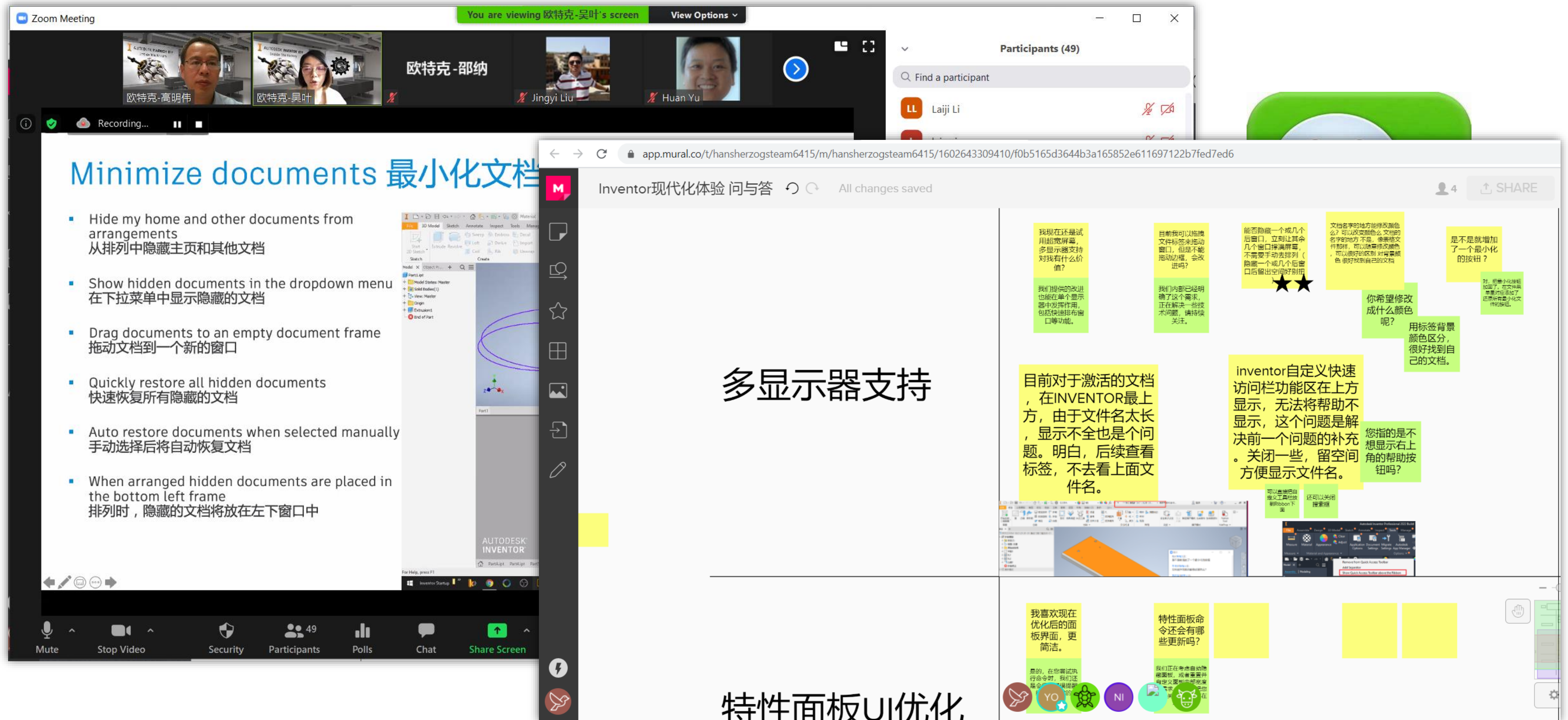
提升用户满意度★







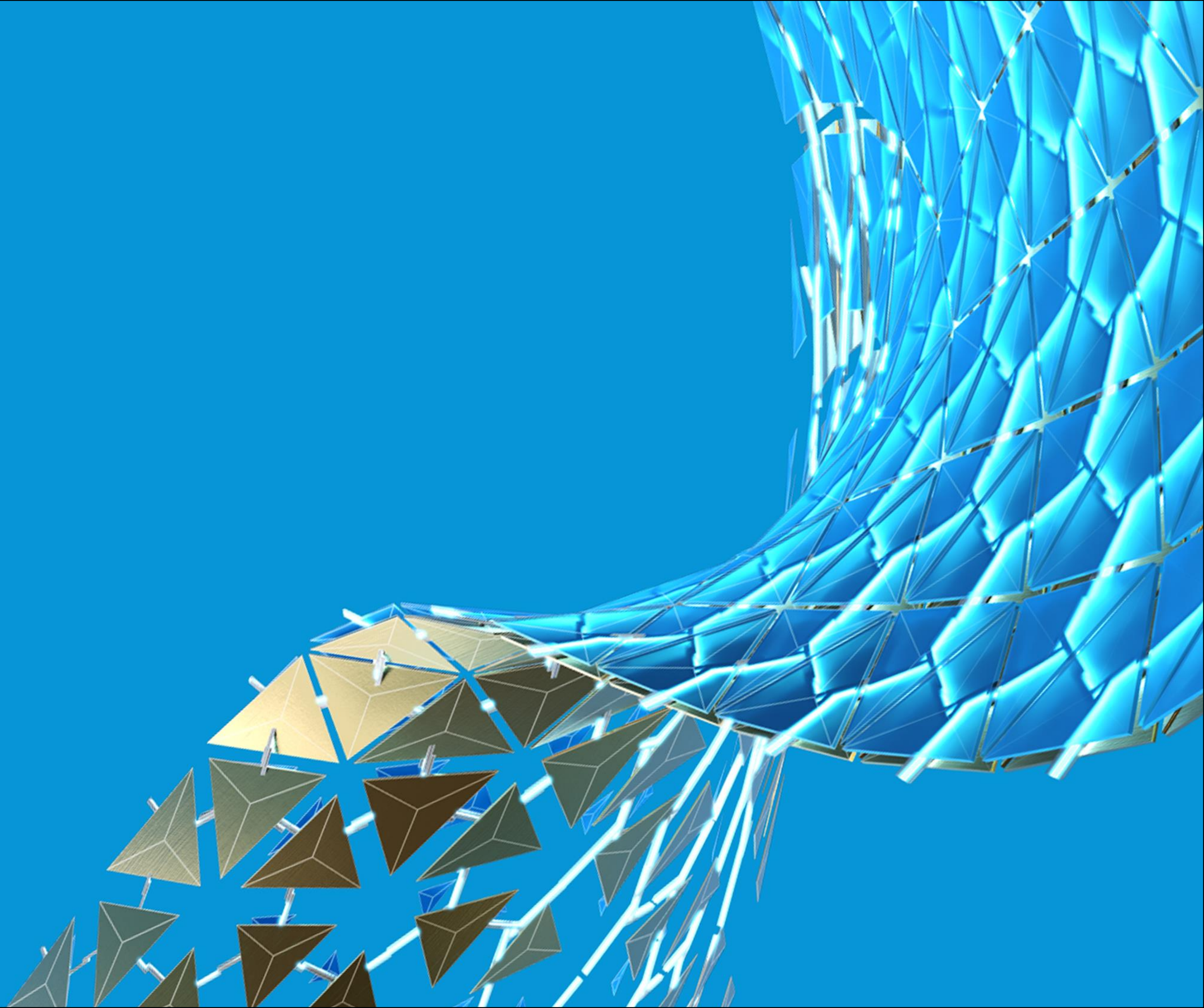
线上沟通成为主要方式





微信ID: autodeskvision

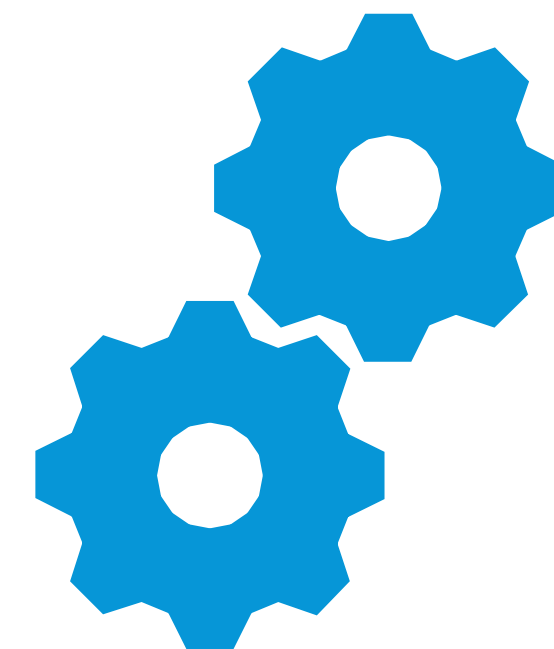
用户互动



通过互动，大家可以...

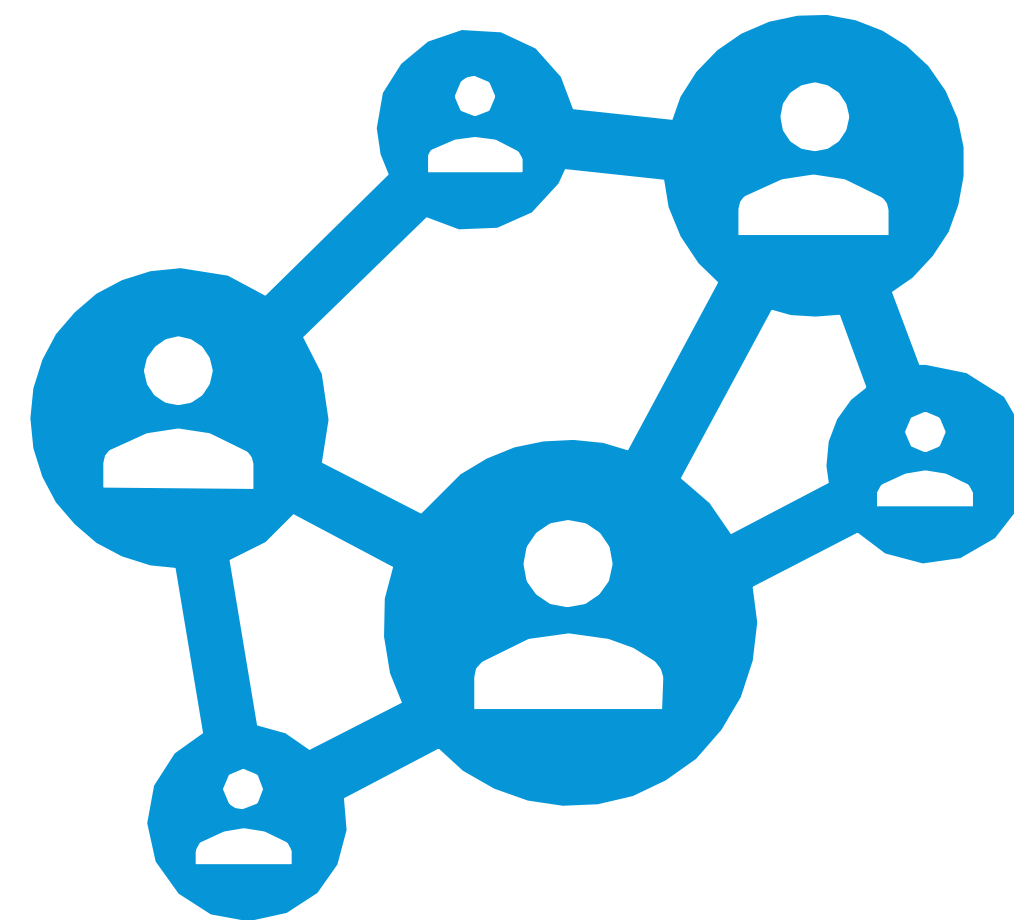
用户

- 寻找解决问题方法；
- 提高产品的应用技能；
- 反馈对产品功能的需求
- 了解产品的技术发展；



产品团队

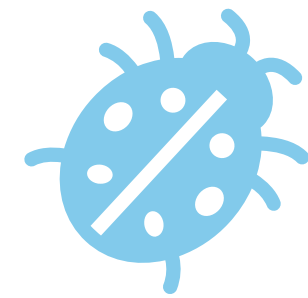
- 理解用户的问题，理清工作流程和影响面；
- 设计验证和功能验证；
- 了解行业的趋势和发展



提升互动效率的建议

产品问题

- 技术支持渠道；
- 产品论坛的问题解答；
- 详尽的问题描述（软件/硬件，必要的步骤）



关键点：

- 是否能够一致性的重新；
- 问题的严重性；
- 有无变通的解决办法；
-

产品需求

- 需要实现的工作流程/目标；
- 必要的说明，帮助了解重要性
- 需求描述（数据，说明以及影响/价值）



关键点：

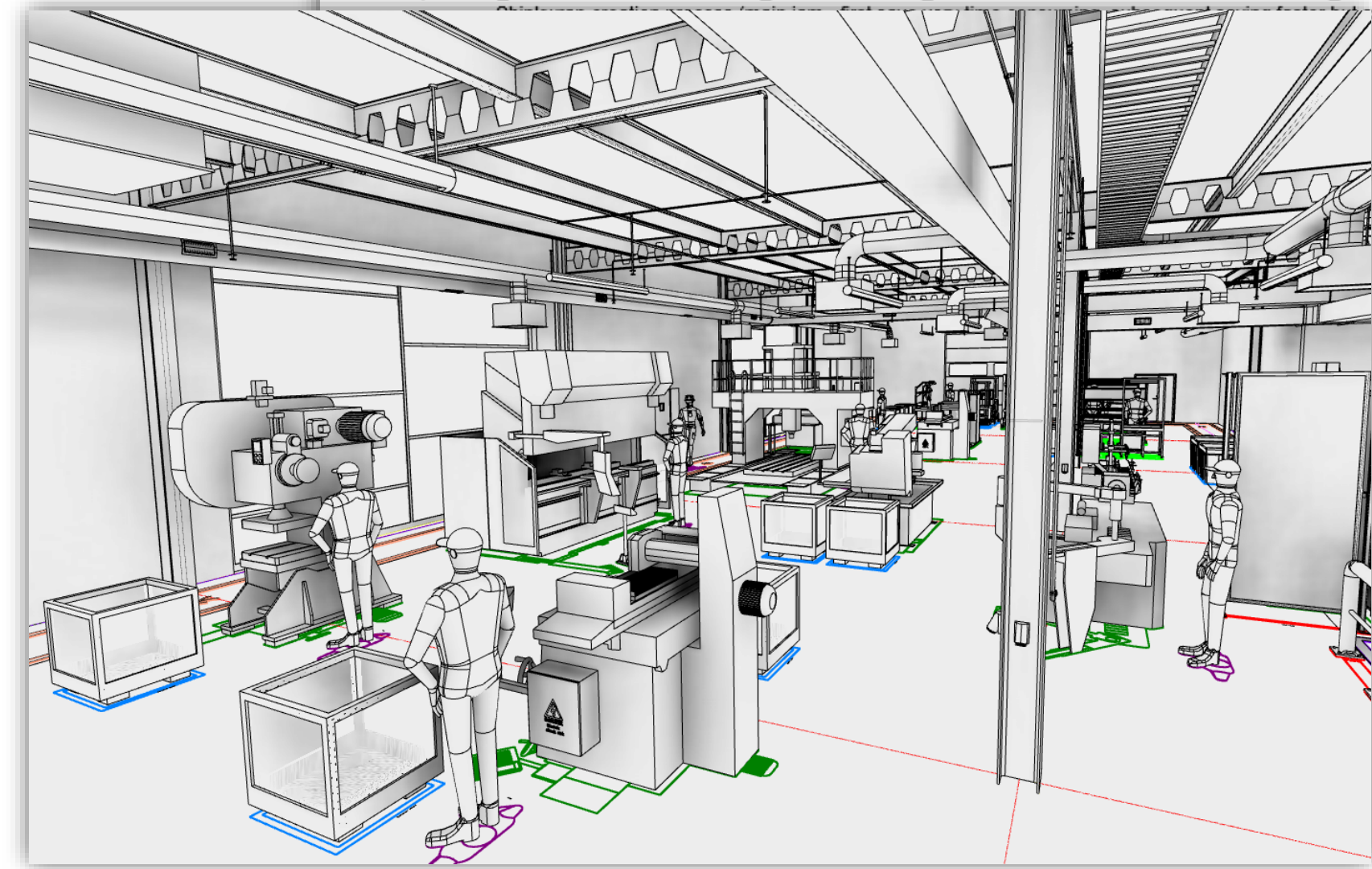
- 需求的普遍性；
- 对用户的影响，行业价值；
- 是否有技术的依赖性；
-

用户互动的建议

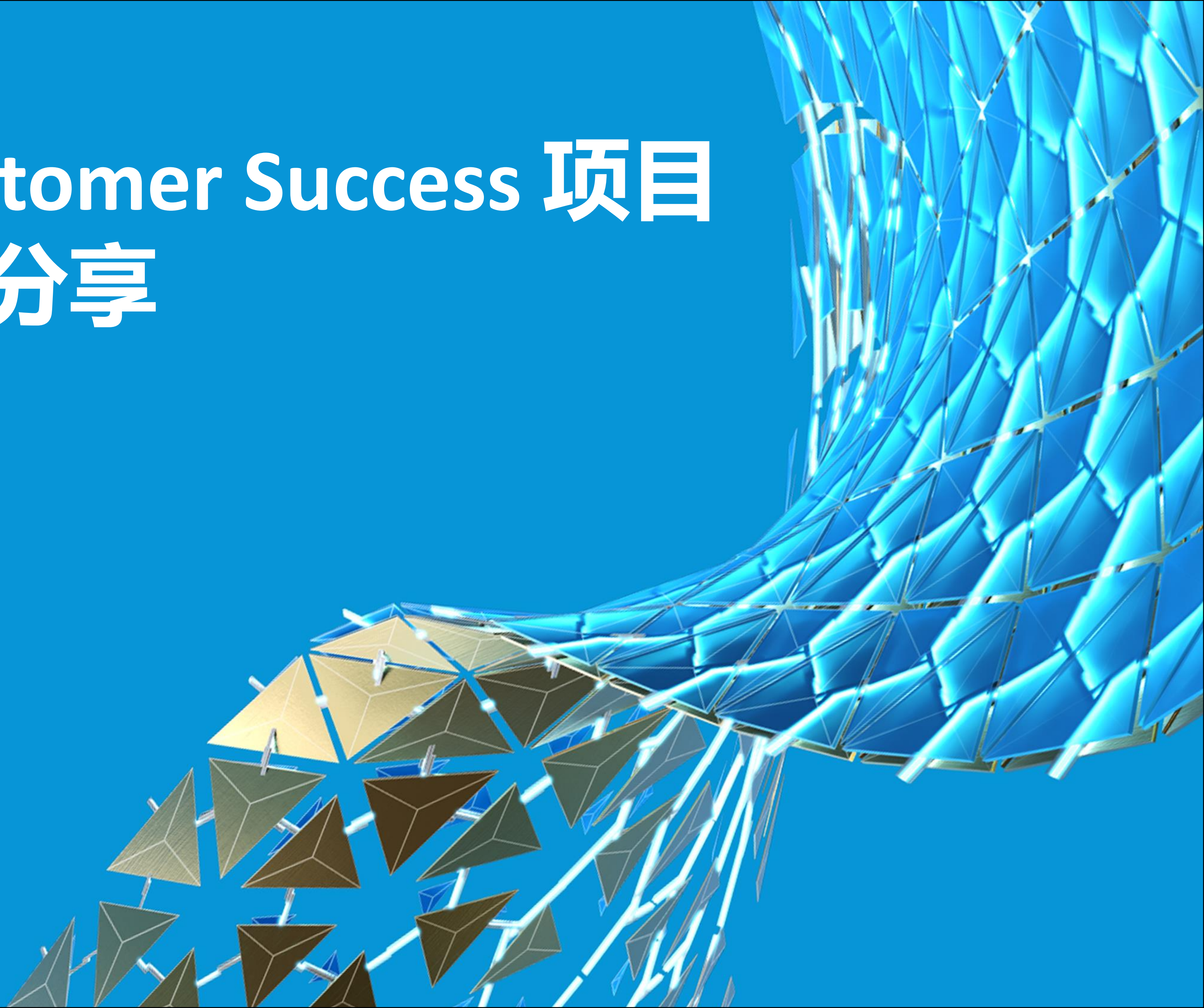
- 基于双方的理解和信任（NDA保密协议）
- 详尽的描述和必要的数据；
- 双方的互动和协力合作；

Performance projects – Assemblies AnyCAD and FDU 2021/2020

- Full LoD model with anyCAD referenced CATIA assemblies Inventor 2021
(\\VAULT_Arbeitsordner\CAD\TH\02\VP_BR46x\MaE\SI\H02_MOPF_SI_Vorrichtungen_FDU_H.iam):
 1. Open model without Vault-Add-In 4:15 minutes (load full 7:30 minutes)
 2. Executing update process very time consuming
 3. Change LoD from <shrink>LoD to Master is very time consuming (maybe Vault-Add-In 2020 problem)
 4. Place Component is very time consuming (maybe Vault-Add-In 2020 problem)
 5. Change LoD of Element is extremely time consuming (maybe Vault-Add-In 2020 problem)
- Full LoD model with anyCAD referenced CATIA assemblies Inventor 2020
(\\VAULT_Arbeitsordner\CAD\TH\02\VP_GMOPF\MaE\SI\H02_MOPF_SI_Vorrichtungen_FDU_H.iam):
 1. Open model Vault-Add-In logged-in up to 45 minutes (load full 14:30 minutes)
 2. Executing update process very time consuming
 3. Change LoD from <shrink>LoD to Master is very time consuming (maybe Vault-Add-In 2020 problem)
 4. Place Component is very time consuming (maybe Vault-Add-In 2020 problem)
 5. Change LoD of Element is extremely time consuming (maybe Vault-Add-In 2020 problem)
- AnyCAD referenced CATIA assembly Inventor 2021
(\\VAULT_Arbeitsordner\CAD\TH\02\VP_BR46x\MaE\Dat_In\EBZ\20190405\SI\TH02DSI11110WZ01000_V00K_ZB_STATION_1110_RP.iam):
 - Shrinkwrap creation process (main iam - first save very time consuming, subsequent saving faster but still too time consuming)
 - Change LoD from Master to <shrink>LoD is time consuming
- AnyCAD referenced CATIA assembly Inventor 2020
(\\VAULT_Arbeitsordner\CAD\TH\02\VP_GMOPF\MaE\Dat_In\EBZ\20190405\SI\TH02DSI11110WZ01000_V00K_ZB_STATION_1110_RP.iam):
 - Shrinkwrap creation process (main iam - first save very time consuming, subsequent saving faster but still too time consuming)
 - Change LoD from Master to <shrink>LoD is time consuming



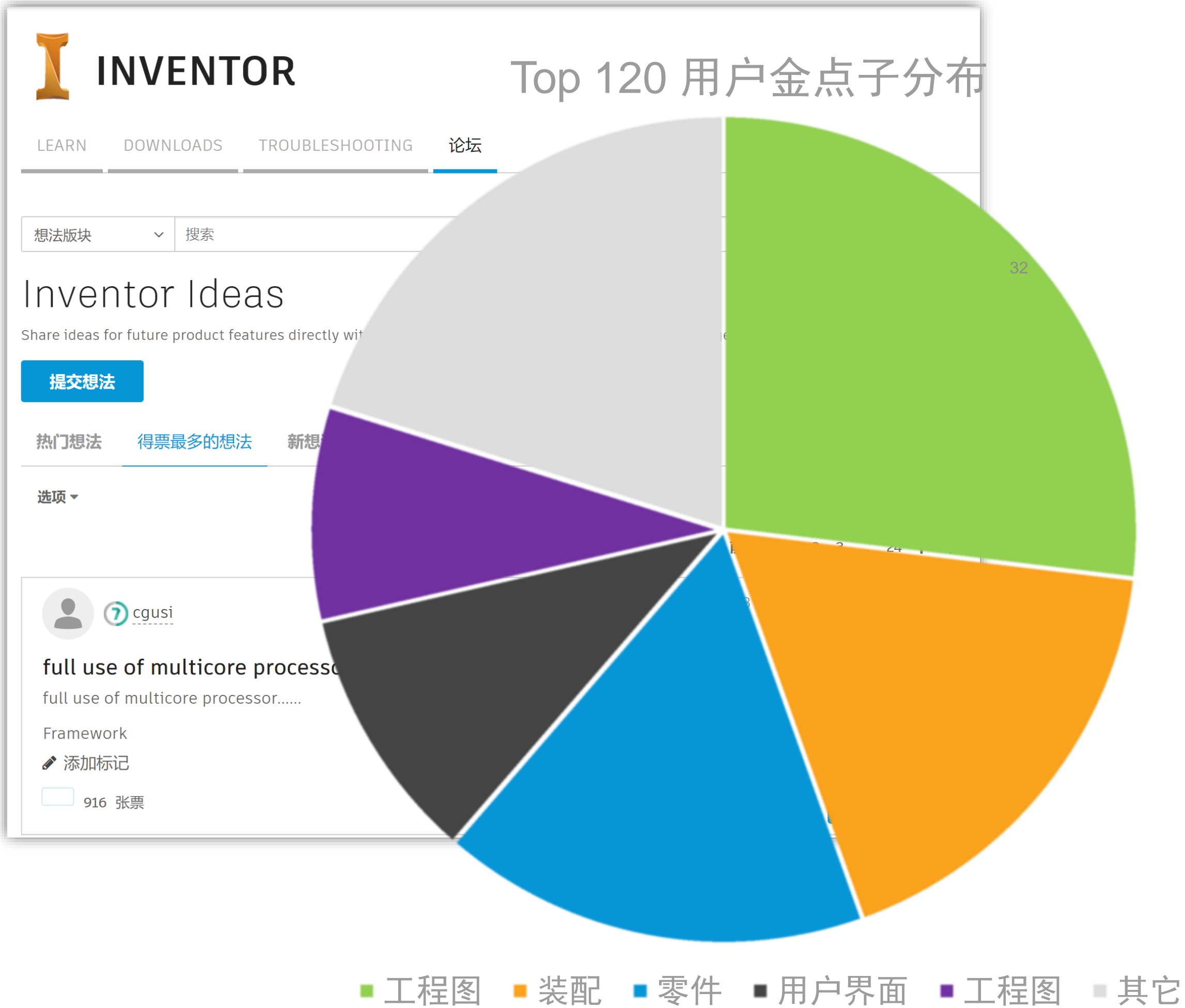
Inventor Customer Success 项目 的部分案例分享

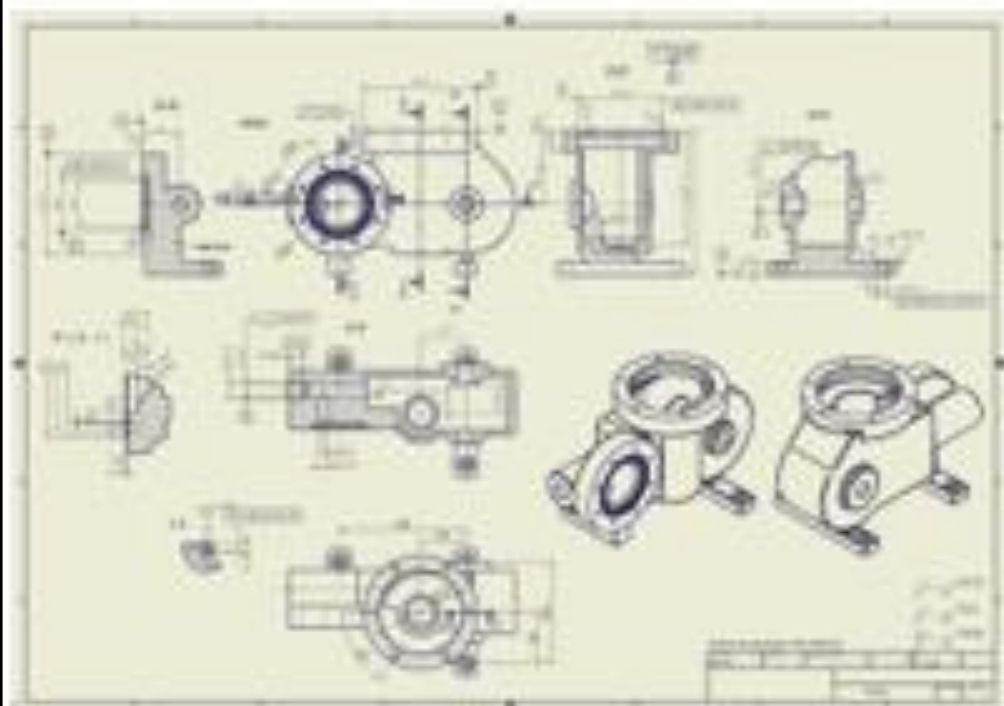
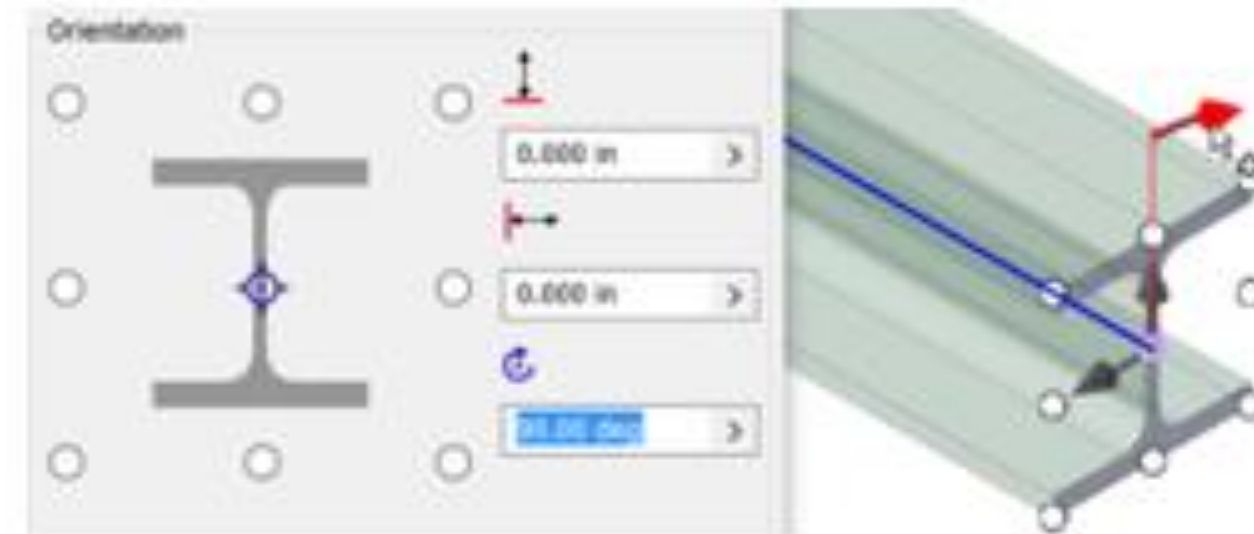
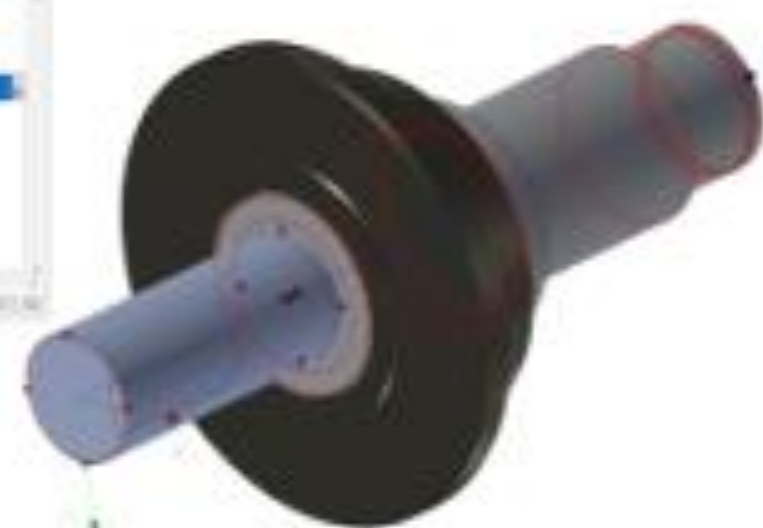


用户需求



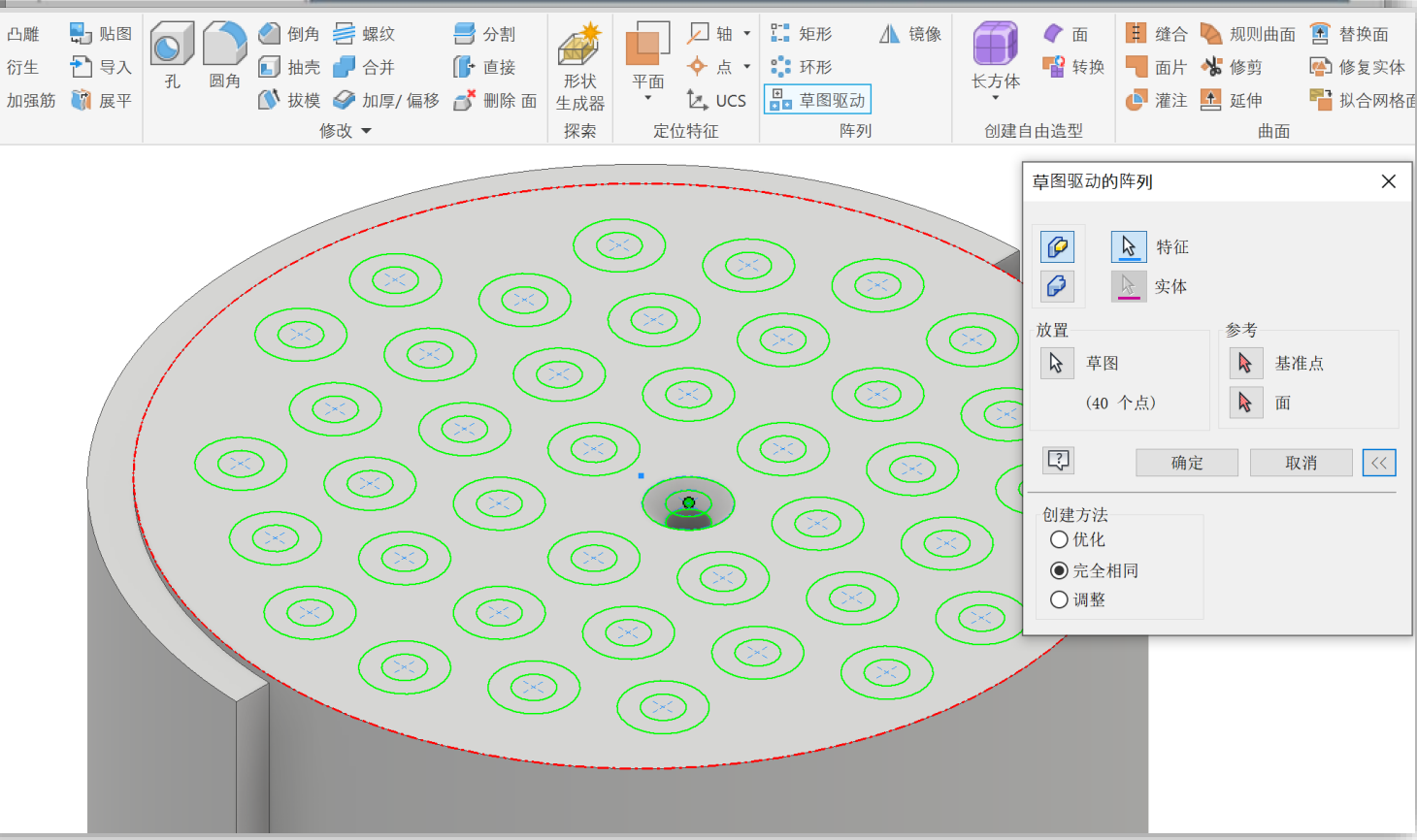
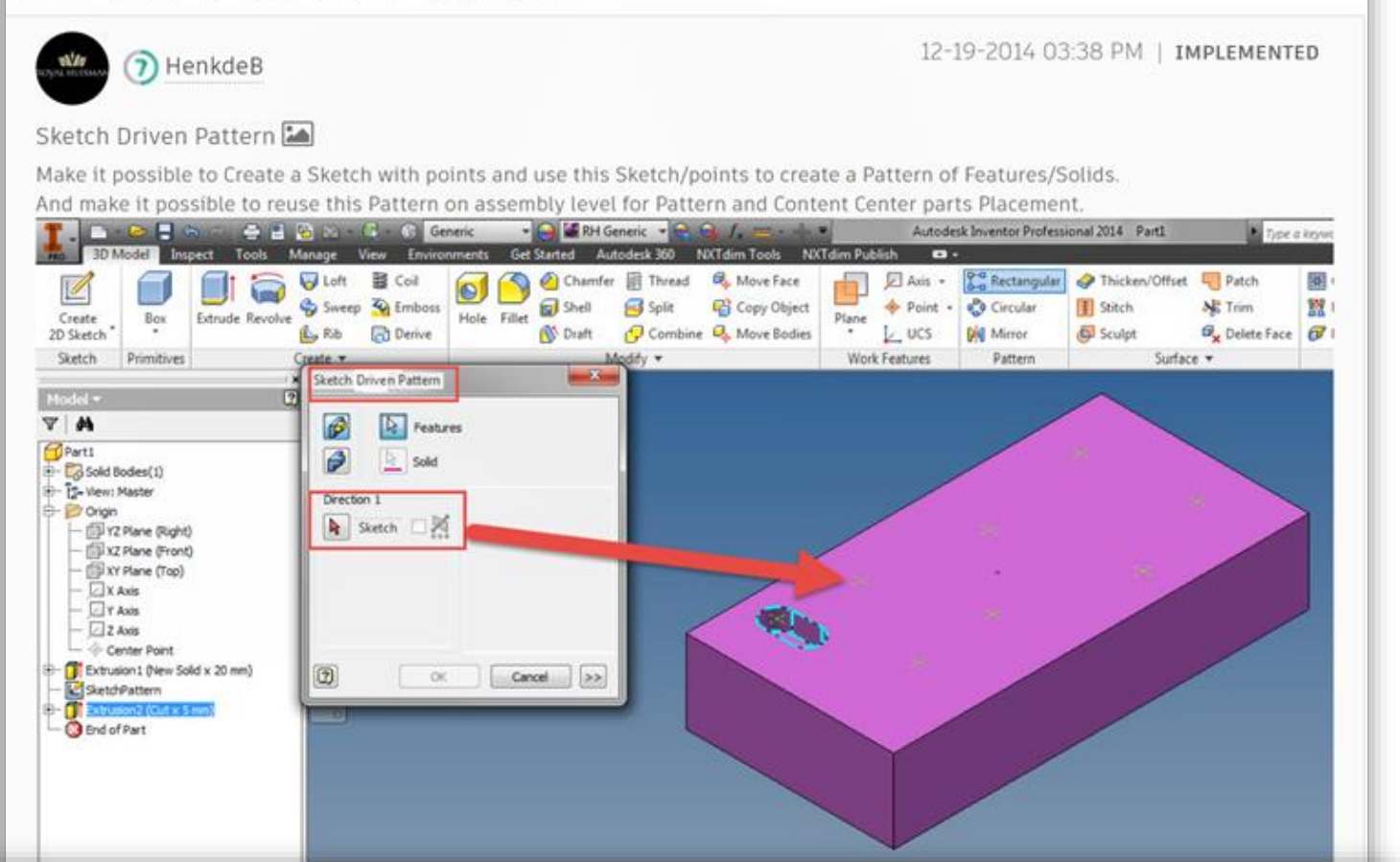
性能
用户金点子
钣金
模型状态表达
结构件
iLogic
建模
表达视图
管路
用户体验



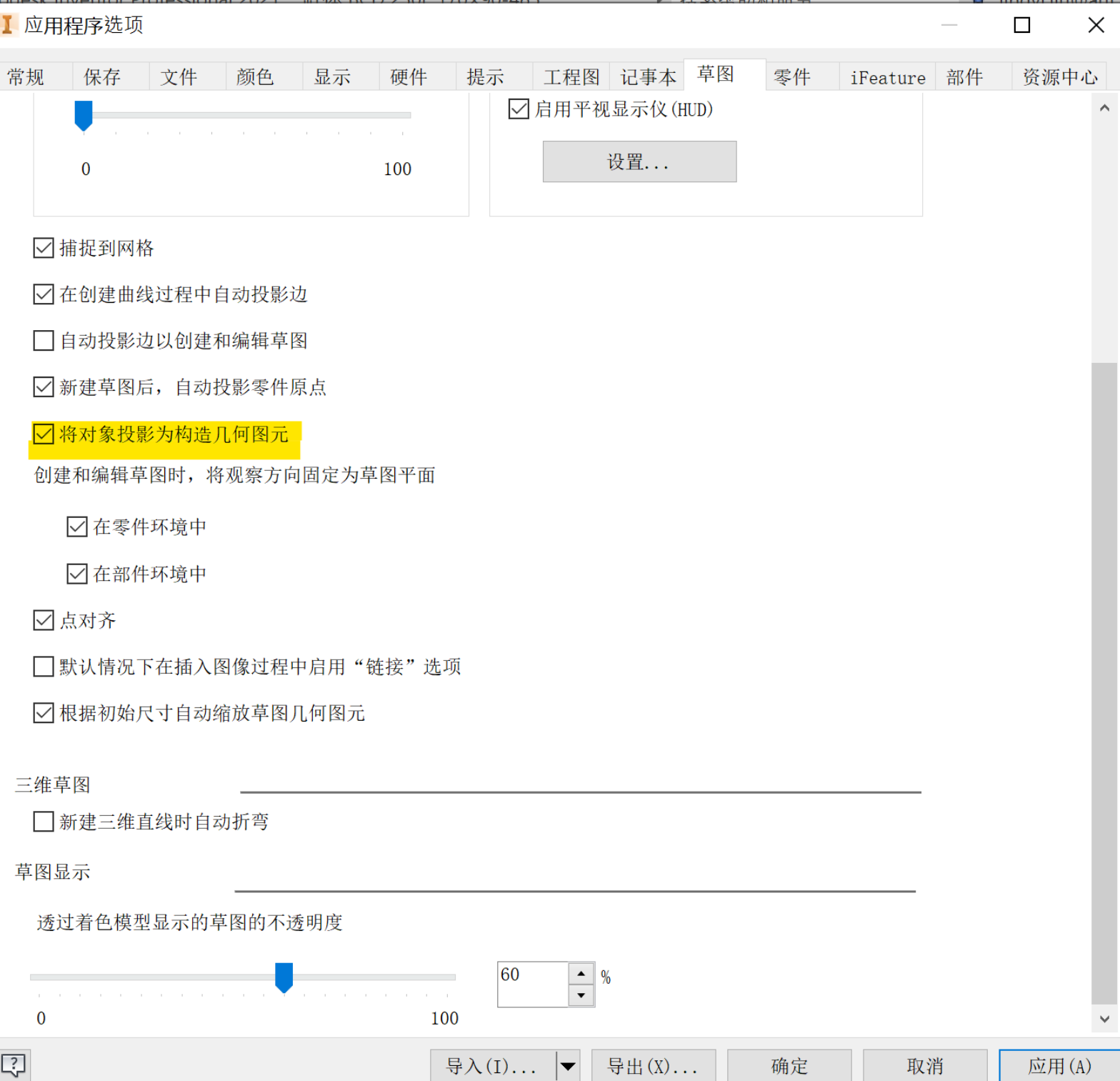
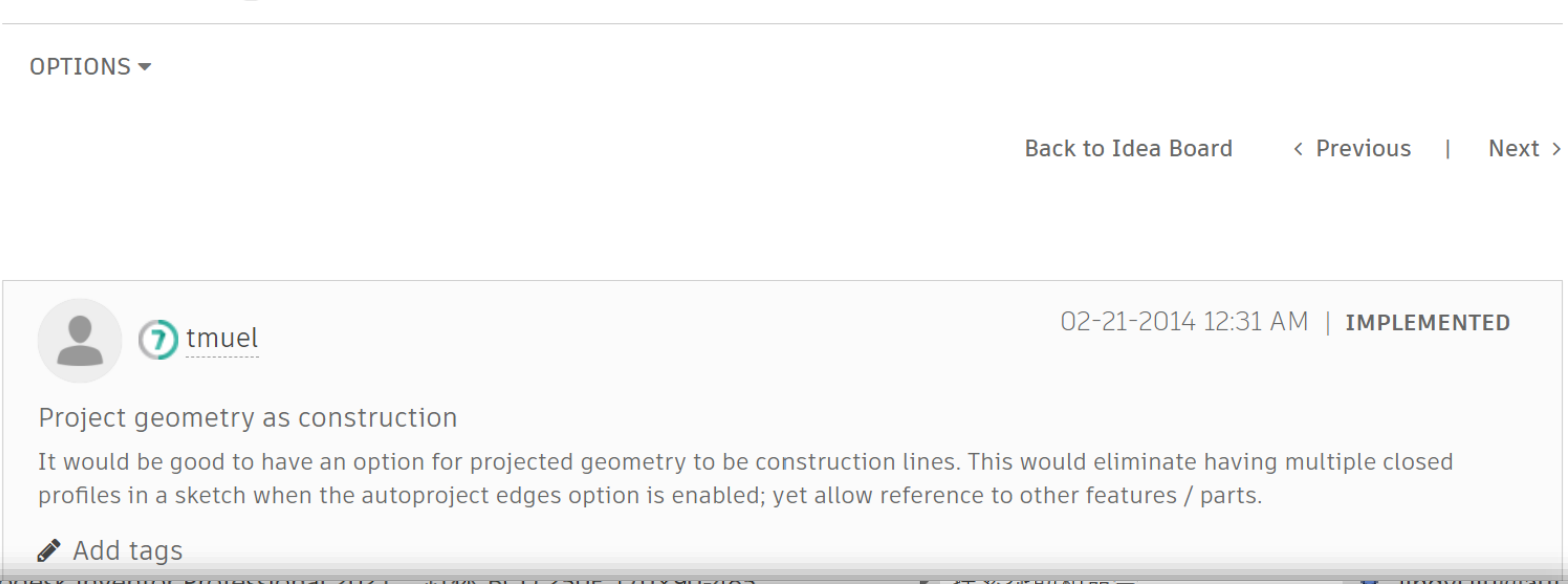


草图

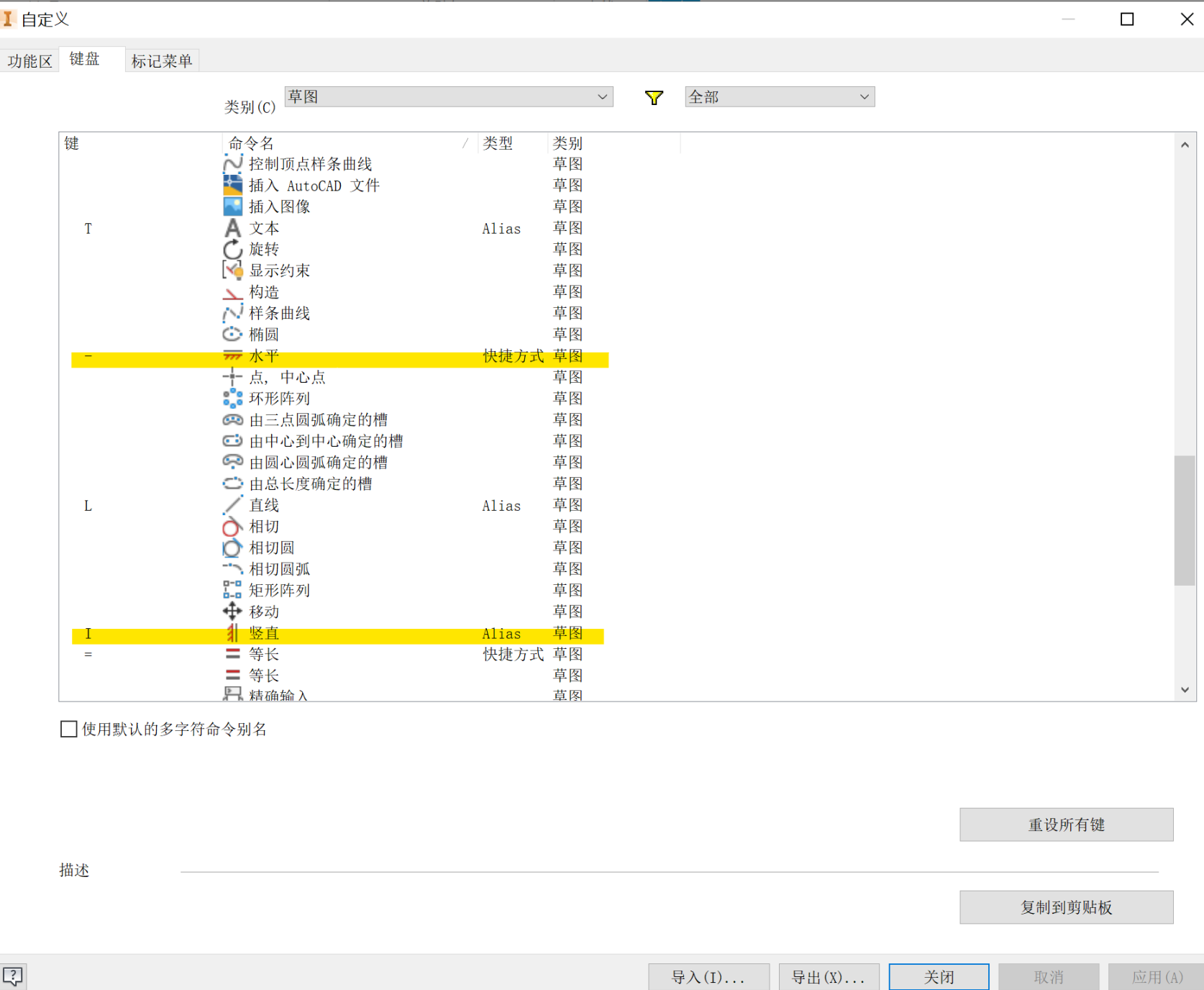
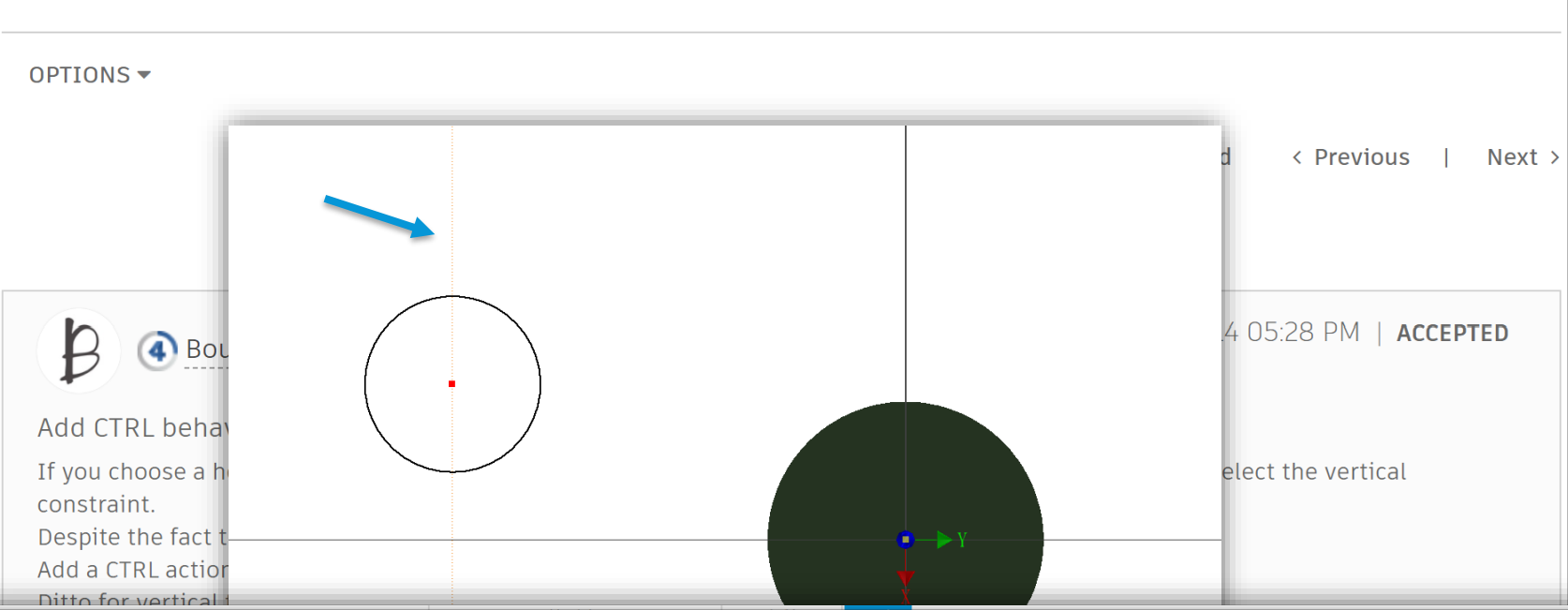
Sketch Driven Pattern



Project geometry as construction



Add CTRL behavior to Horizontal and Vertical sketch constraint.



实体扫掠

New Feature Sweep solid

OPTIONS ▾

[Back to Idea Board](#) < [Previous](#) | [Next](#) >



11 Marco.Takx

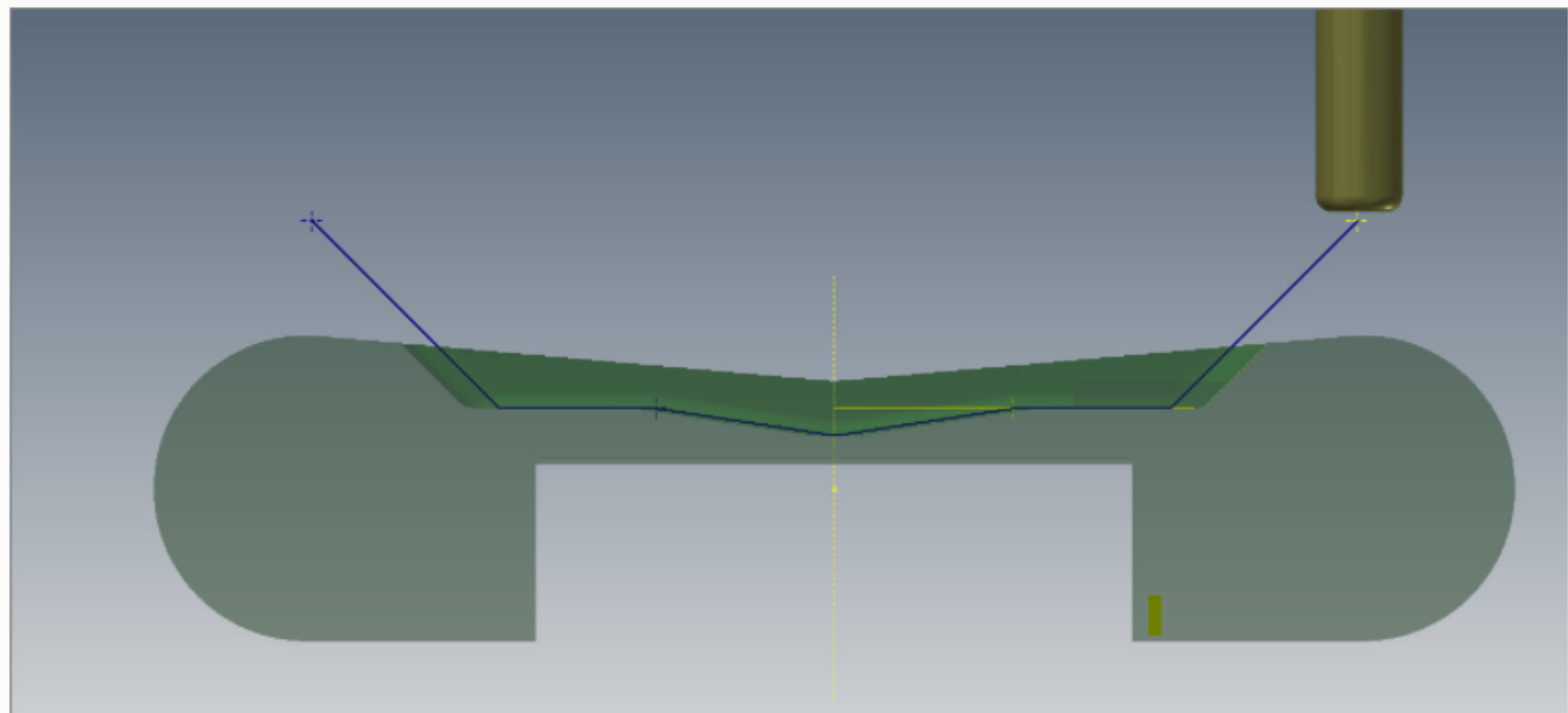
10-13-2014 05:32 PM | **IMPLEMENTED**

New Feature Sweep solid

Hi Autodesk,

Now we have Autodesk Inventor HSM, it will be nice to have a new feature that sweeps a solid along a path. When you use a profile milling it is hard to design the result. But when you simulate it within Inventor HSM, you see how the results look.

So please make a feature to design easier results like this.



cad CAM, inventor, Inventor HSM, Milling, new feature, products, Sweep

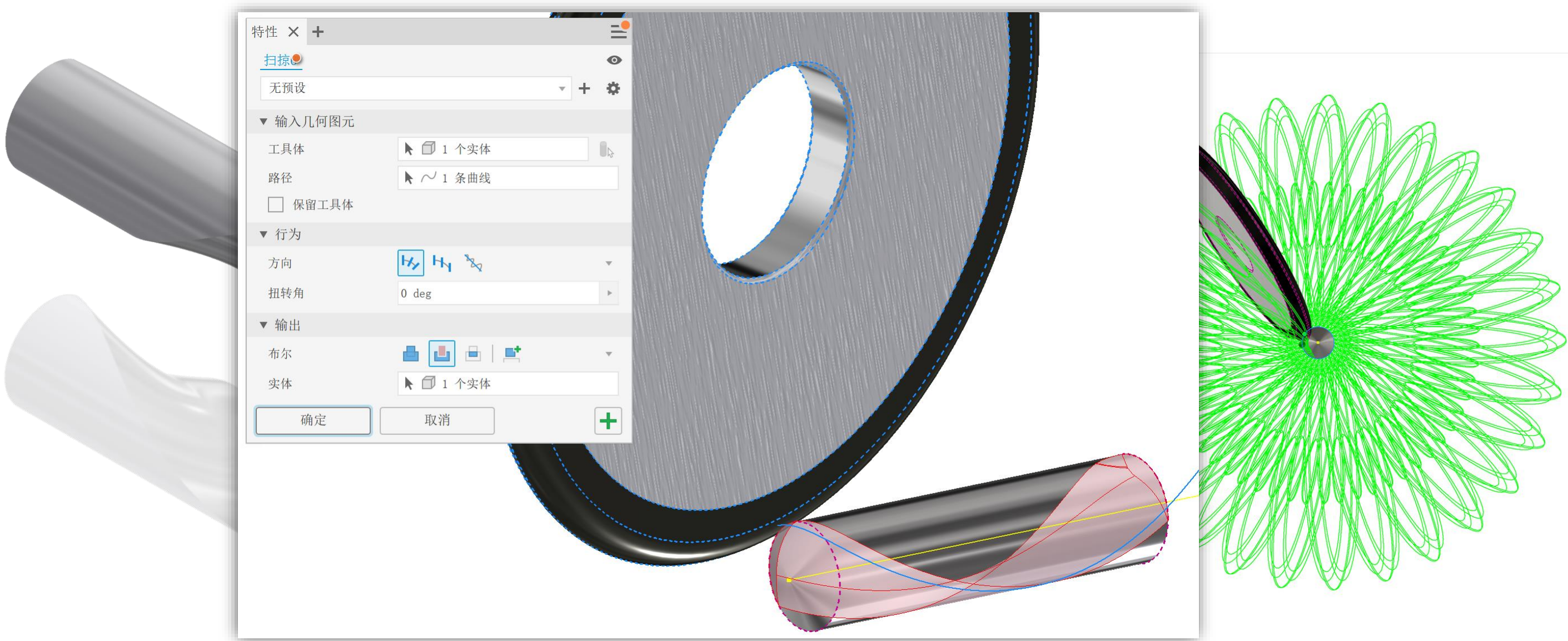
Report

Add tags

[VOTE](#) 609 Votes

63 Comments (13 New)

[POST A COMMENT](#)



干涉检查

Thread interference analysis filters

OPTIONS ▾



Thread interference analysis filters

Good morning, I suggest to make possible to check/uncheck a flag to exclude/include in interference analysis...

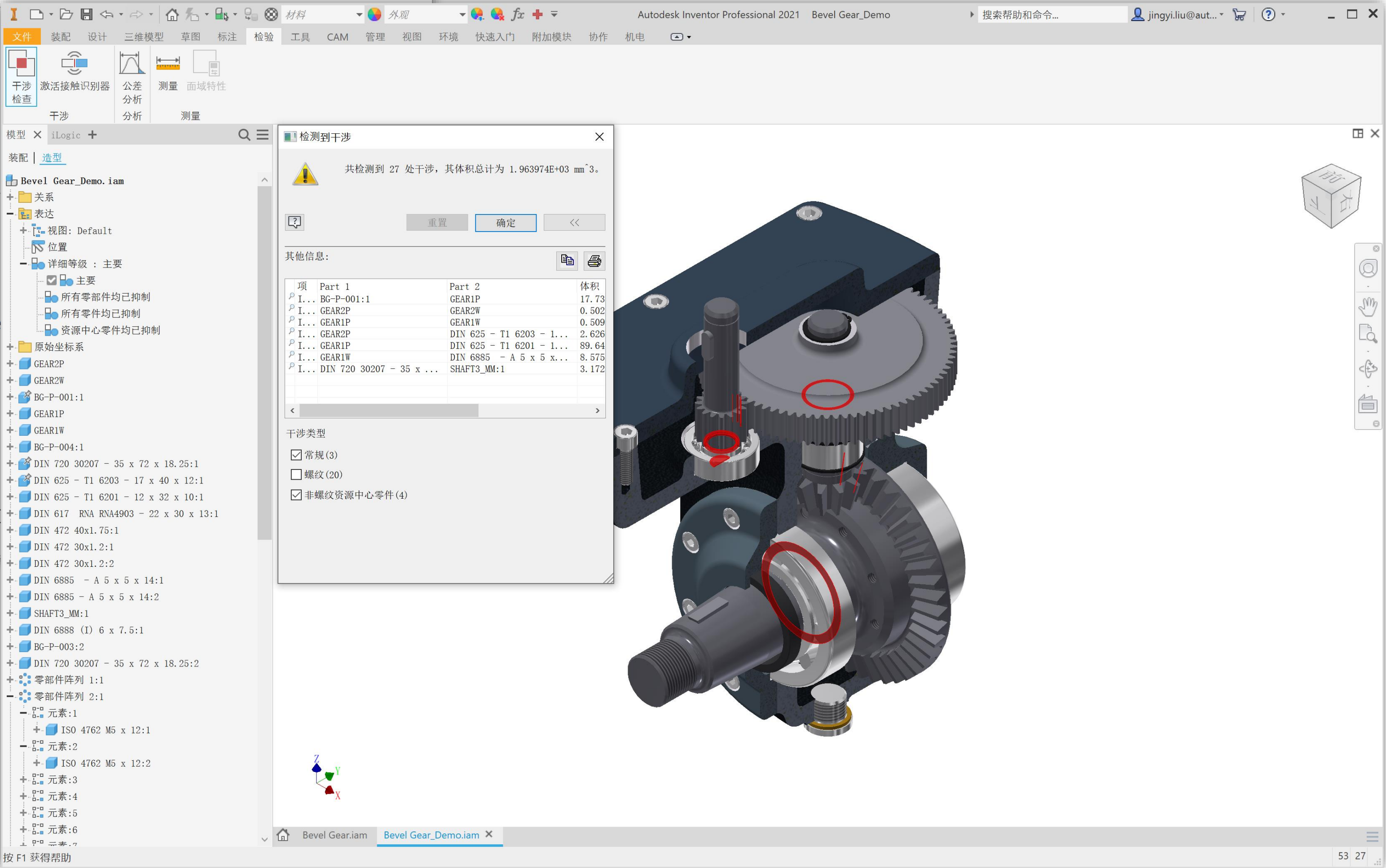
This option could simplify the interference analysis of assembly...

Thanks

Add tags

VOTE 77 Votes

180



工程图

Smarter Diameter Dimensions

OPTIONS ▾

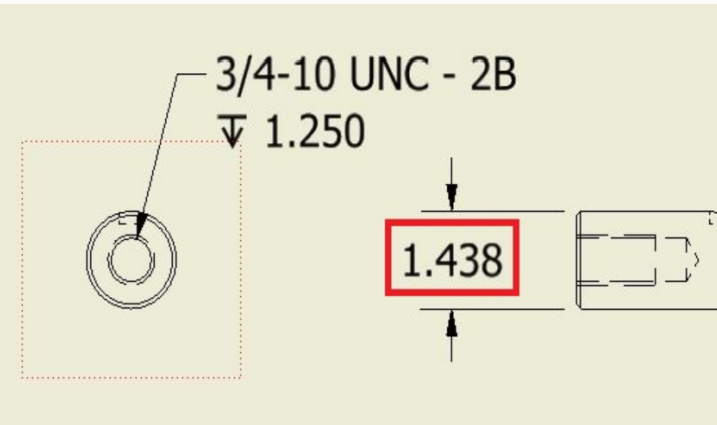
Back to Idea Board < Previous | Next >

mrattray

04-05-2013 01:46 AM | IMPLEMENTED

Smarter Diameter Dimensions

Shouldn't Inventor know that this is a cylinder and should therefore have a diameter symbol? Let's have it pop in there on its own without me having to do an edit text operation on it and manually add it.



Drawings

Add tags

VOTE

773 Votes

71 Comments (51 New)

POST A COMMENT

▲ Report

Custom alignment direction for Aligned Dimensions

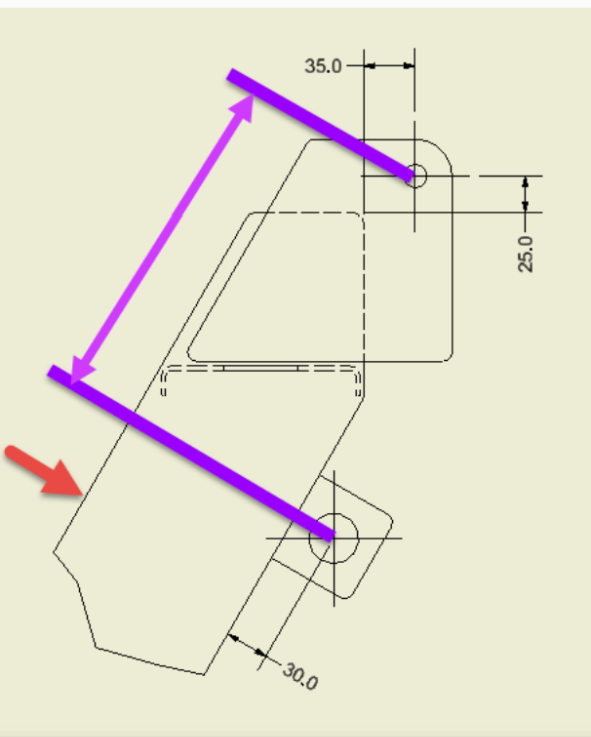
brendan.henderson

10-06-2014 05:13 AM | IMPLEMENTED

Custom alignment direction for Aligned Dimensions

I added [this post](#) to the Inventor forum hoping I was missing something. It appears I'm not.

There needs to be better controls to place aligned dimensions. I mentioned in the original post that a UCS change (like in Autocad) could do it but I now feel that Inventor should just have a RMB option to select an edge to align the dimension to.



DETAIL A

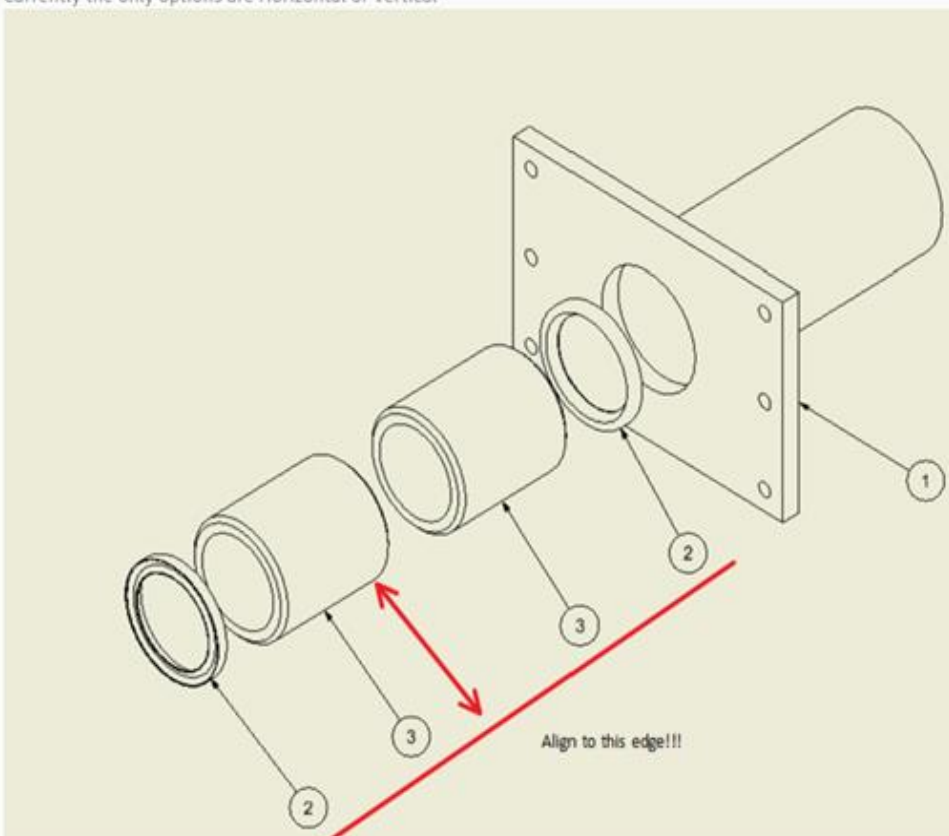
Align Balloons to an Edge

exSWX

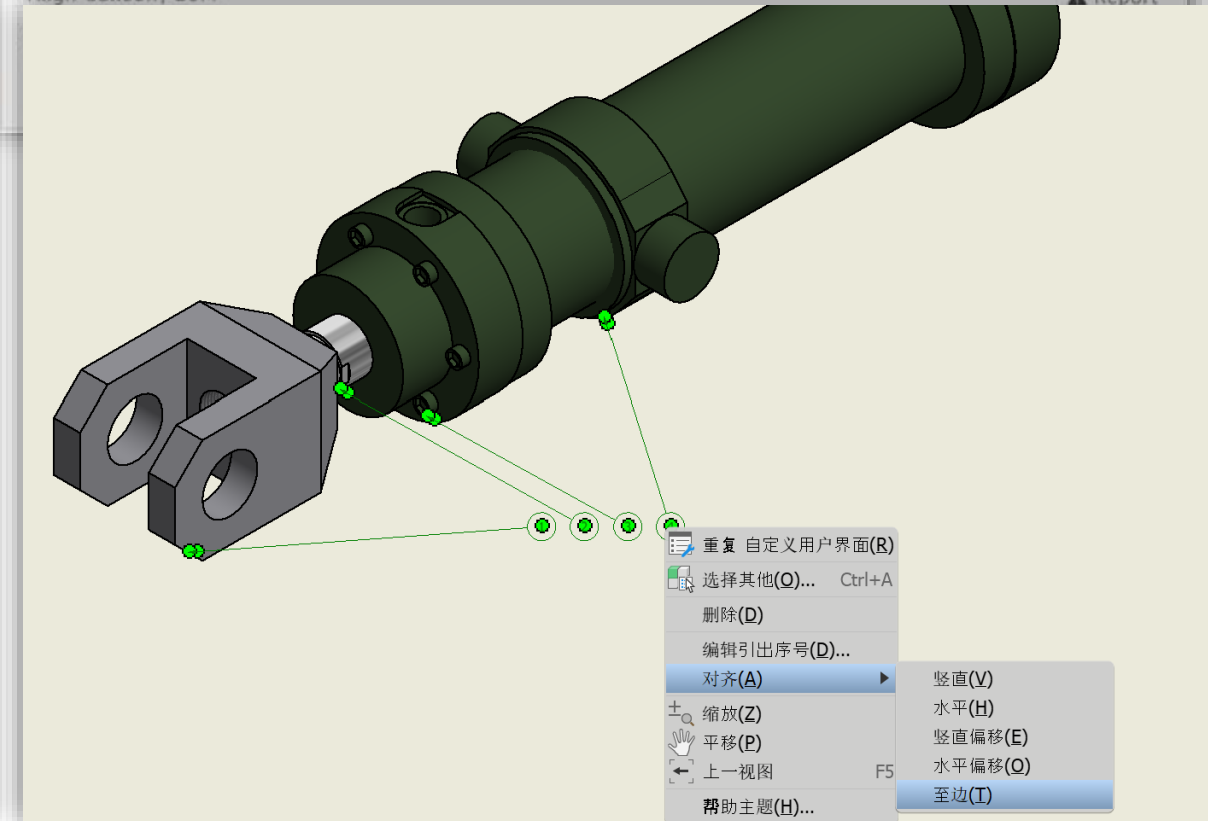
11-11-2014 10:50 AM | IMPLEMENTED

Align Balloons to an Edge

I'd like the ability to align the balloons on a drawing to be aligned to an edge. Currently the only options are Horizontal or Vertical



Align_balloon_BoM



Reference Parts and Shaded Drawing Views

mthomas

04-30-2015 05:34 AM | IMPLEMENTED

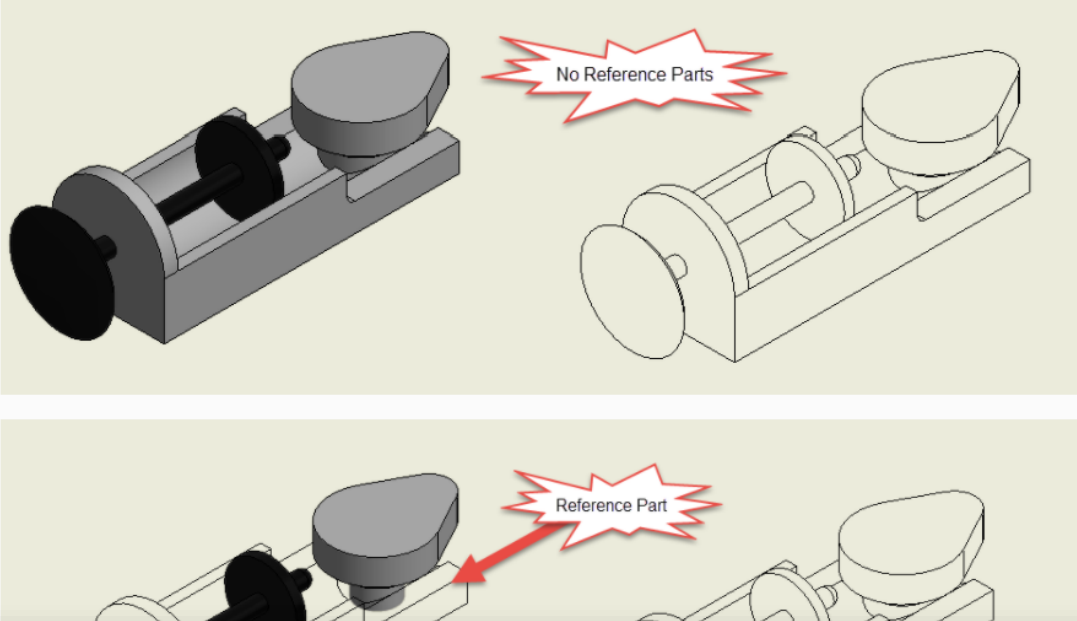
Reference Parts and Shaded Drawing Views

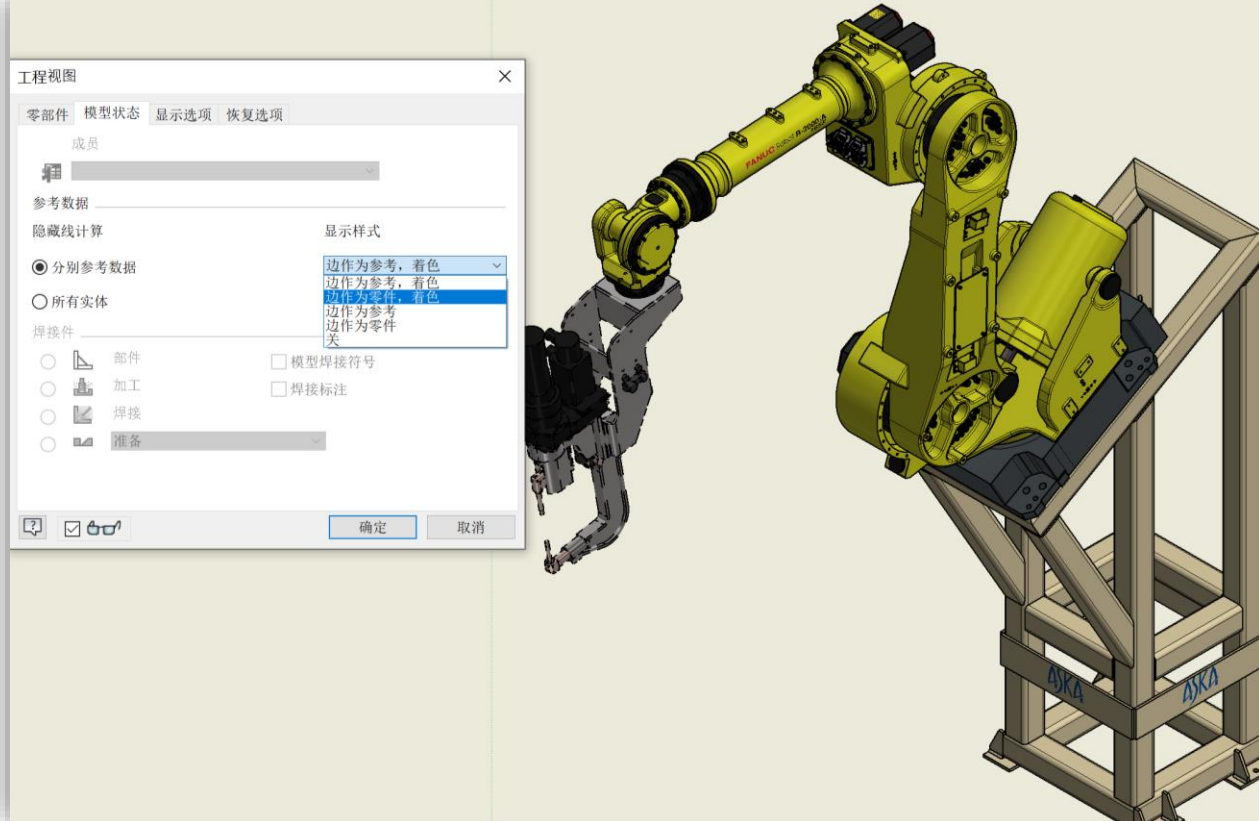
Reference parts and shaded drawings views do not play nice together.

#1 Why can I not shade reference parts in a drawing view? There should be four Line Style options: As Reference Parts, As Parts (Non-Shaded), As Parts, and Off. With *As Parts* it will use the style of the view... i.e. if the view is shaded then the parts are shaded, if the view isn't shaded then the parts are not shaded.

#2 When the view is shaded but the reference parts are not, the reference parts "bleed through" the shading and do not show proper hidden line / line removal. This makes interpreting the view very difficult.

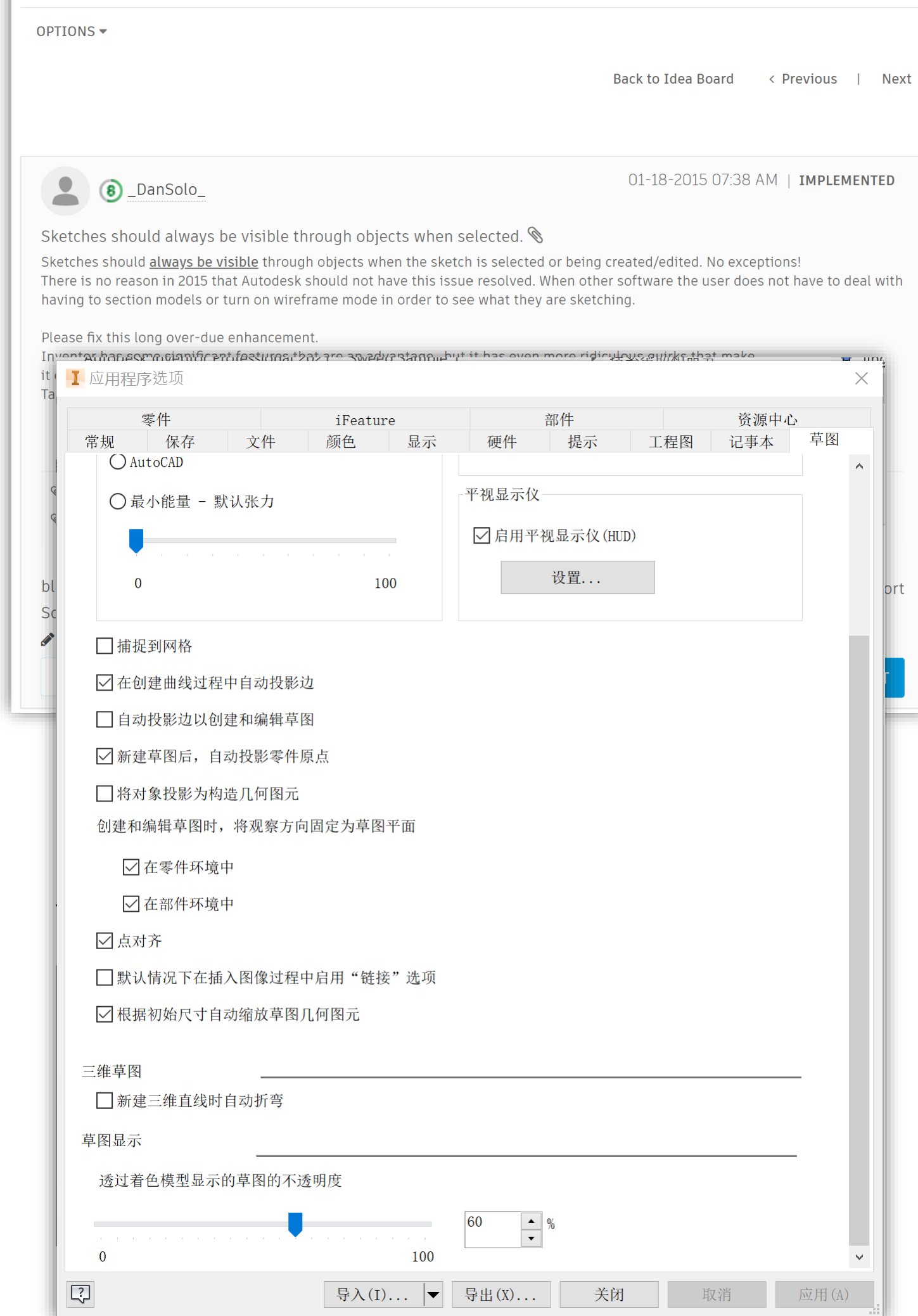
In the first image, with no reference parts, the views look good. However, notice in the second image where you cannot tell that the Valve component goes through a hole. It seems to be floating on top of the assembly.



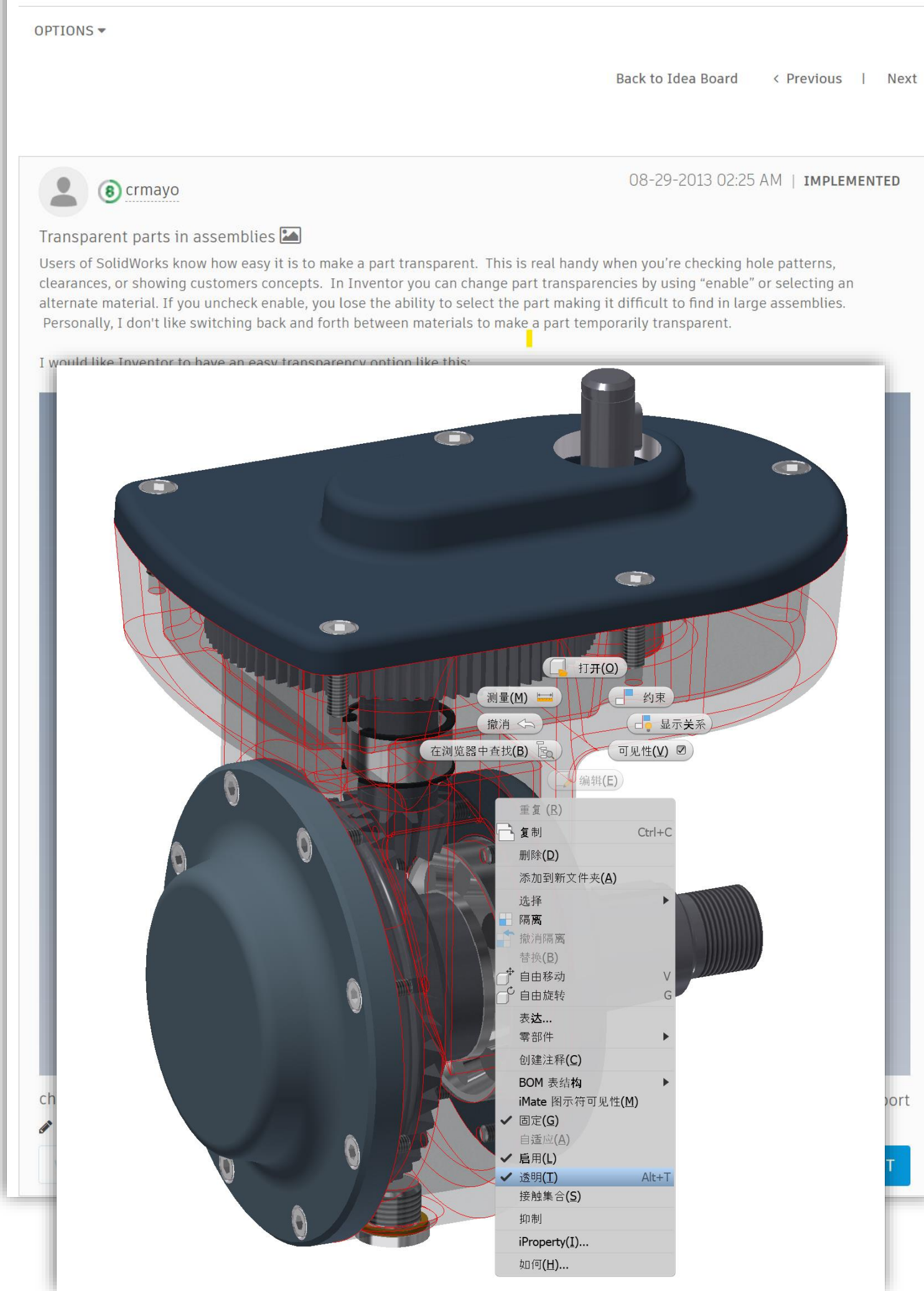


透明设置

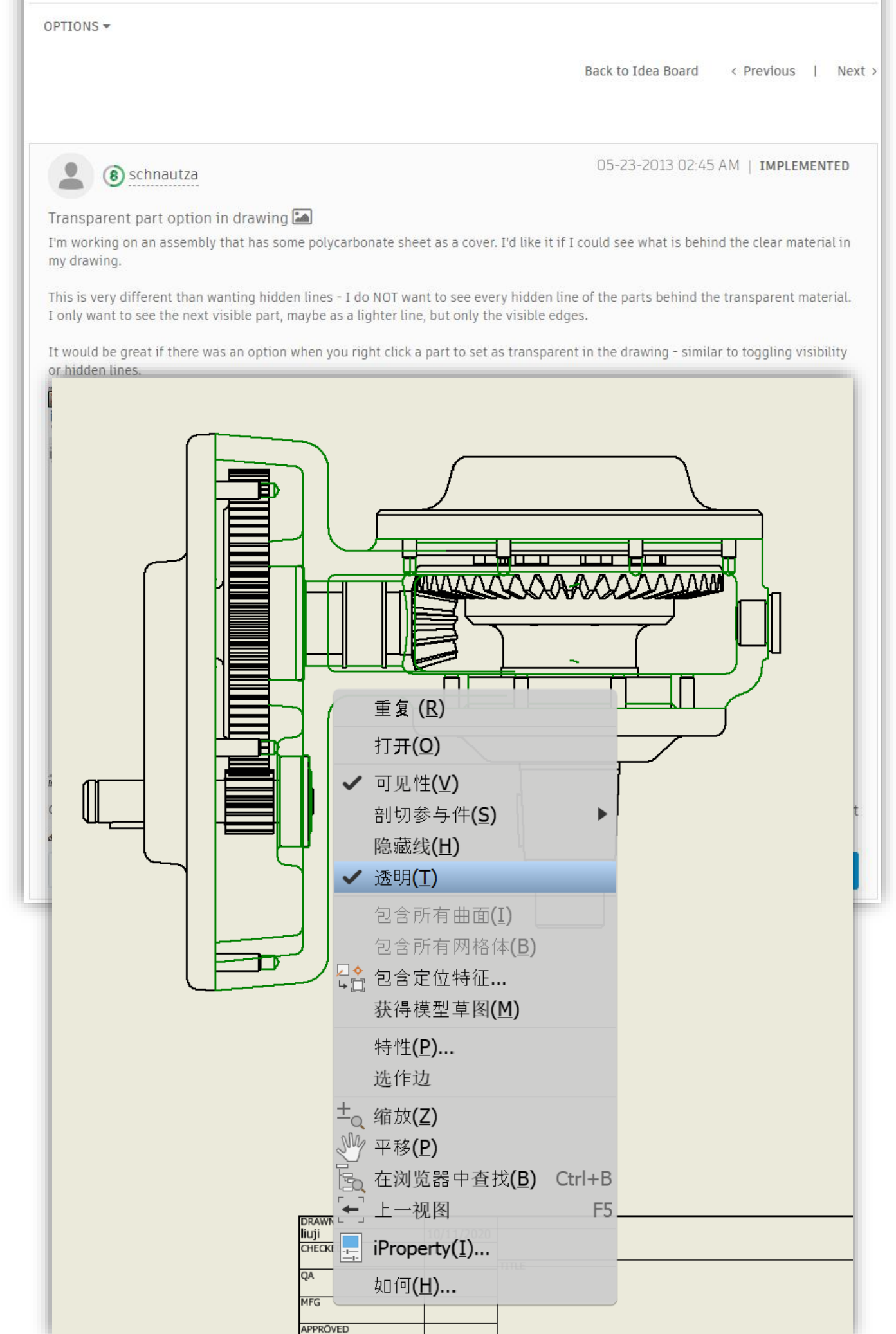
Sketches should always be visible through objects when selected.



Transparent parts in assemblies



Transparent part option in drawing



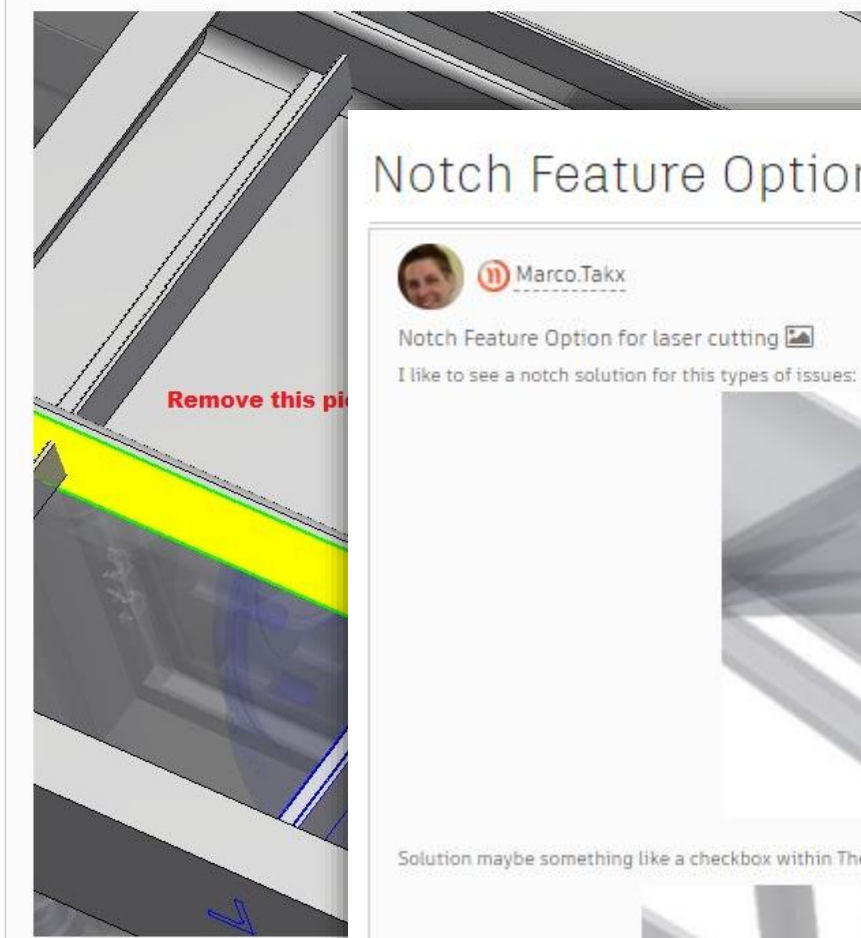
结构件生成器

Frame Generator Trim - Select Side

10-09-2013 01:42 AM | IMPLEMENTED

Frame Generator Trim - Select Side

It would be nice in Frame Generator, when trimming, to be prompted for which end of the member you want to keep, or remove. As it is, FG assumes you want the longer segment based on the trim faces selected, and this is not always the case. The work around is very tedious and time consuming.



frame generator

Add tags

VOTE 69 Votes

Notch Feature Option for laser cutting

04-17-2014 10:29 PM | IMPLEMENTED

Notch Feature Option for laser cutting

I like to see a notch solution for this types of issues:

Solution maybe something like a checkbox within The Notch feature.



frame generator

Add tags

VOTE 69 Votes

Frame Generator - Notch With A Gap

OPTIONS

Back to Idea Board < Previous | Next >

11-14-2012 09:37 PM | IMPLEMENTED

Easy end caps in frame generator

OPTIONS

Back to Idea Board < Previous | Next >

06-11-2013 10:12 PM | IMPLEMENTED

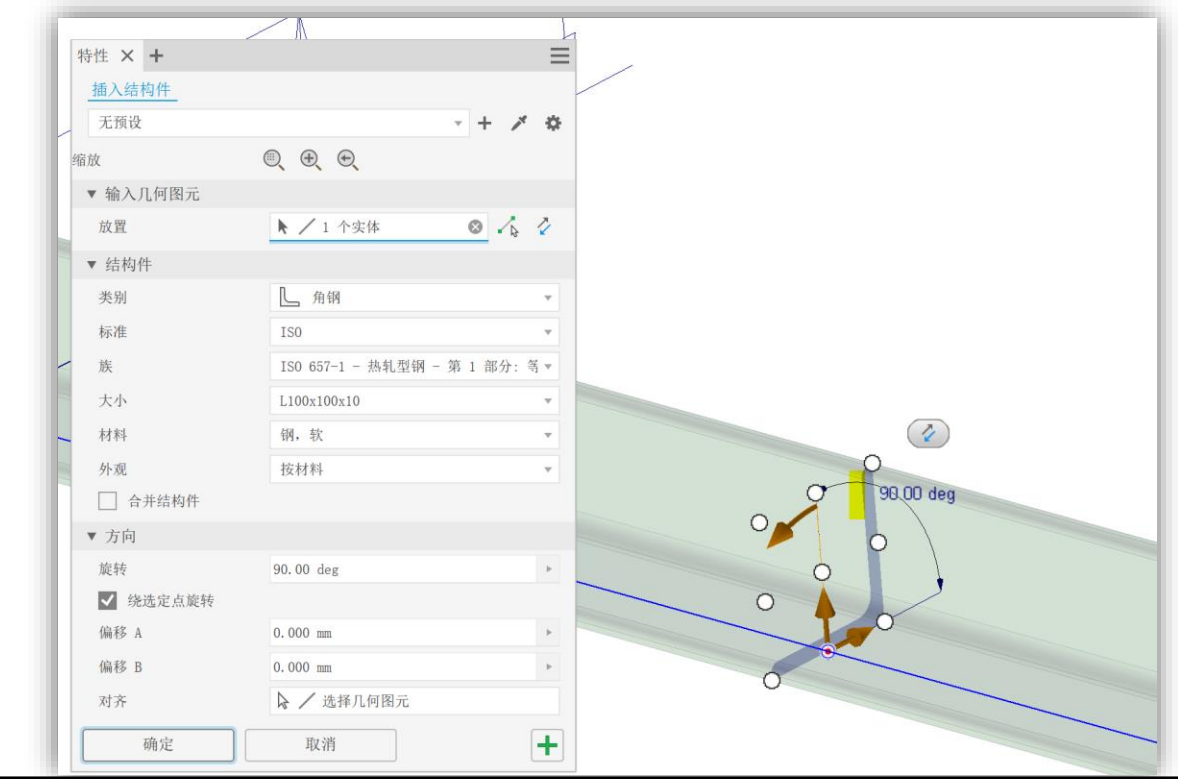
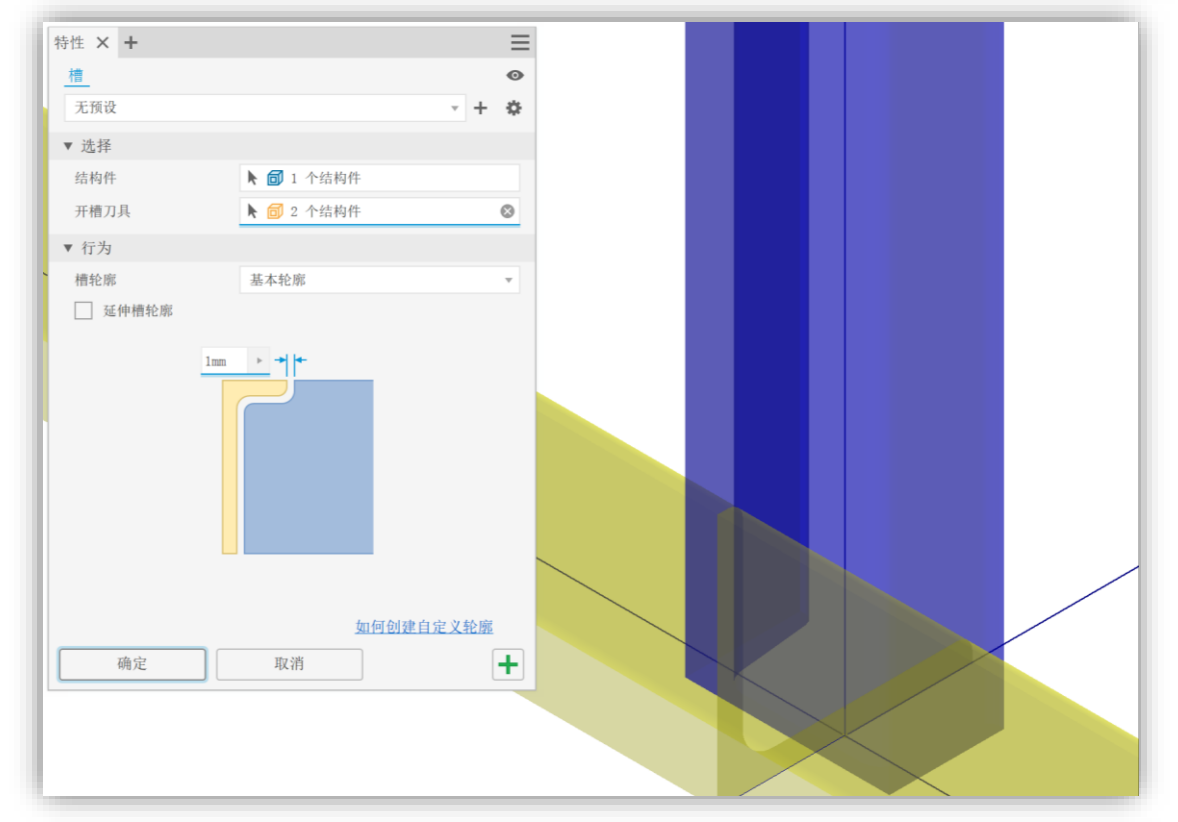
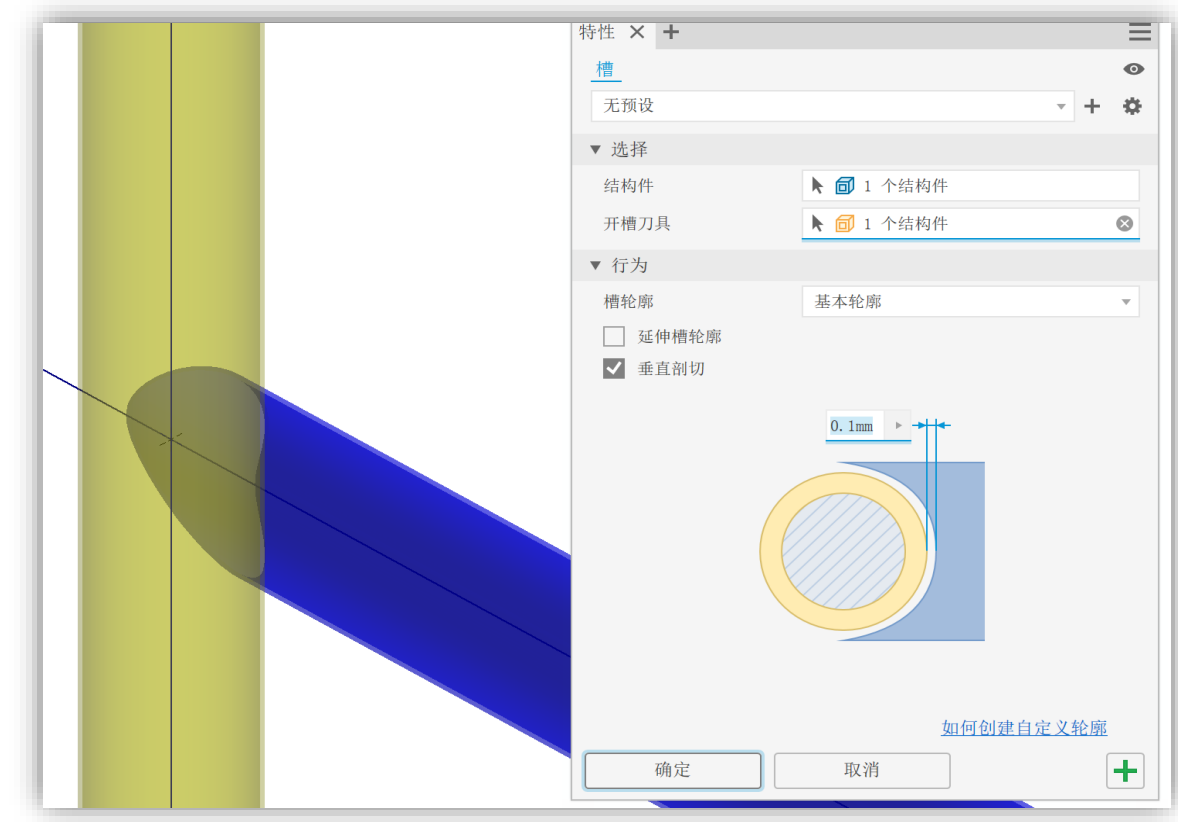
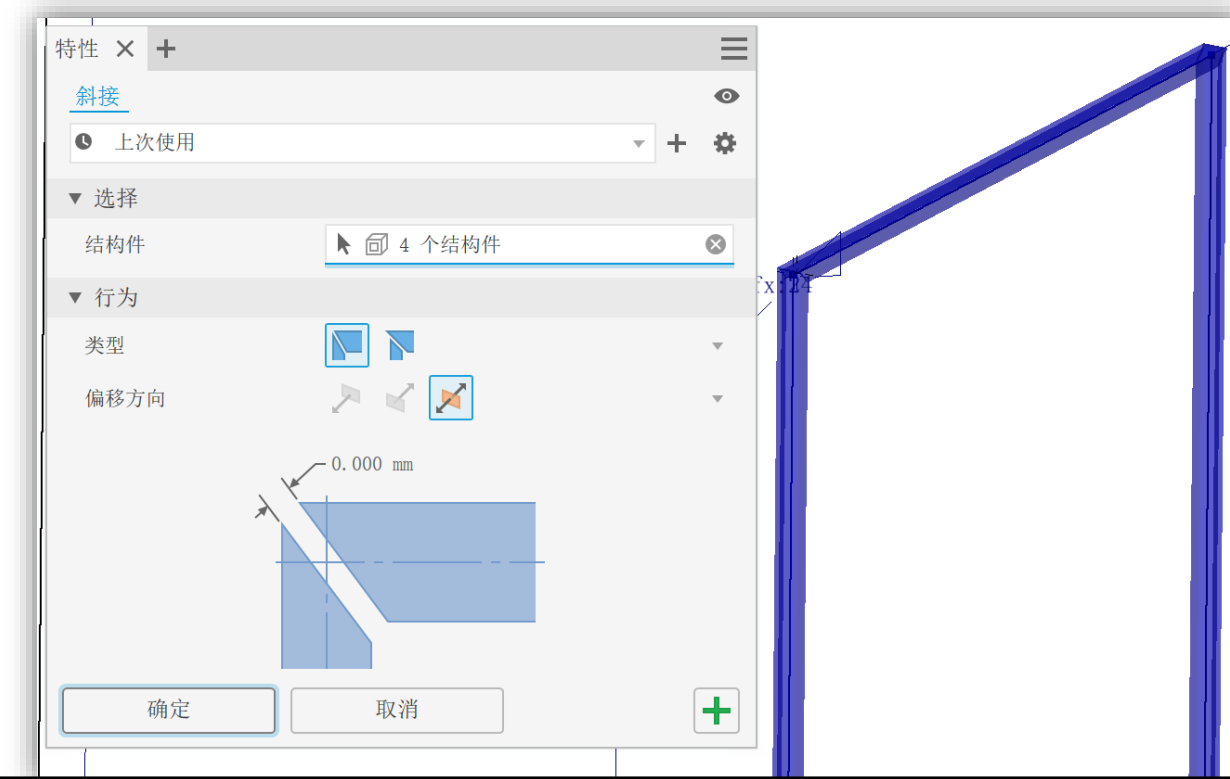
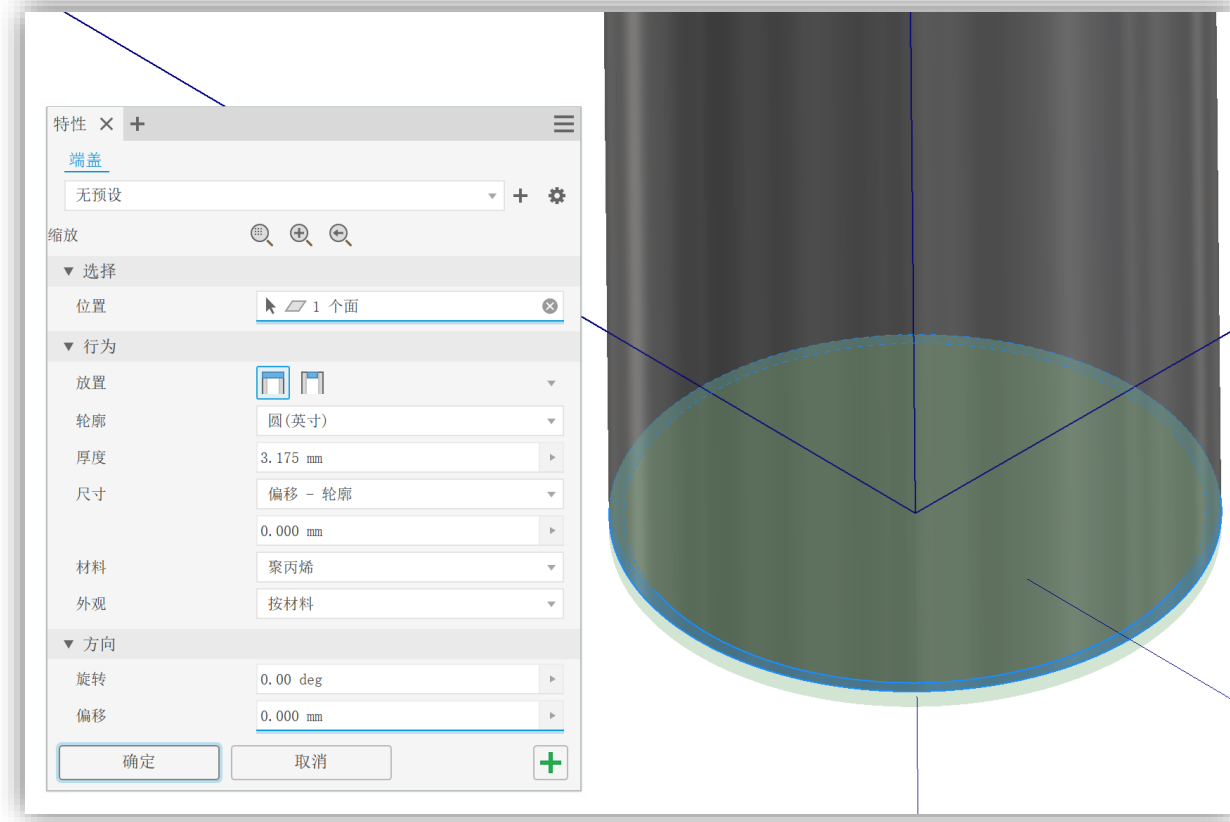
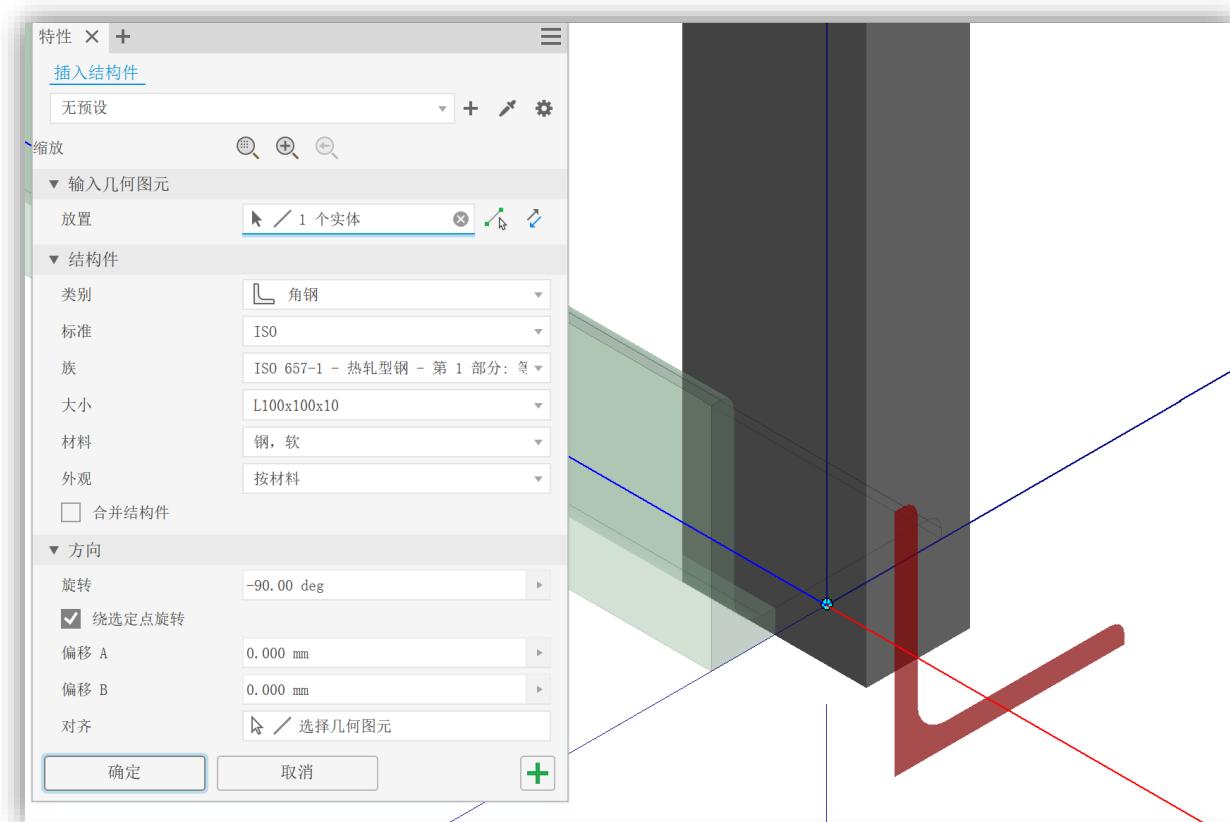
Easy end caps in frame generator

Add a functionality to create end caps very quickly for frames from generator.

Add tags


VOTE 88 Votes

23 Comments (9 New) POST A COMMENT



文件命名

Frame Generator Member Naming

**cbenner**

01-10-2013 03:00 AM | IMPLEMENTED

Frame Generator Member Naming

I think it would be nice, on a per project basis, to have the ability to set a file naming convention for all new frame members in that project.

Currently we name all frame members the same, and those names come from the COnent Center, but then I RENAME them as they are being placed, so that they include the job number as a prefix. That way I avoid having duplicate filenames in Vault, AND... more importantly, it makes the so much easier to copy using Copy Design. I can simply do a find and replace from the old job number to the new job number... very quick if the job number is there.

So... it would be nice to have some sort of box where I could specify a prefix to be added to all new parts generated by FG was they are placed.... it gets a bit tedious renaming hundreds of pieces of steel at a time.

frame generator

Report

Add tags

VOTE 122 Votes


26 Comments (14 New)

POST A COMMENT

Inventor Bolted Connections File Naming Defaults Addition

OPTIONS ▾

Back to Idea Board < Previous | Next >

**jeff.perez**

08-02-2019 04:39 AM | GATHERING SUPPORT

Inventor Bolted Connections File Naming Defaults Addition

I would like to see the implementation of adding Inventor Bolted Connections to the file naming default section in Application Options similar to how Frame Generator and Tube & Pipe are currently being handled in Inventor 2019 & 2020. By default, when adding bolted connections into an assembly model they take the name as "Bolted Connection: 1, 2, 3, etc". Having users manually determine or input a file name for every bolted connection can be cumbersome, especially for assemblies that may have a large amount of these bolted connections. If users have the ability to input a specific index number in the file naming defaults, would solve the potential unique file naming issues with vault.

Bolted Connection Inventor Ideas.png

application options bolted connections, frame generator, Tube&Pipe

Report

Add tags

VOTE 61 Votes

6 Comments (6 New)

POST A COMMENT

应用程序选项

常规 保存 文件 颜色 显示 硬件 提示 工程图 记事本 草图 零件 iFeature 部件 资源中心

撤销

%USERPROFILE%\AppData\Local\Temp\1\

默认模板

配置默认模板...

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Tem

设计数据(样式等)

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Des

预设

%USERPROFILE%\AppData\Roaming\Autodesk\Inventor %

草图符号库文件夹

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Des

默认资源中心文件

%USERPROFILE%\Documents\Inventor\Content Center F

项目文件夹

%USERPROFILE%\Documents\Inventor\Projects\

默认 VBA 项目

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Mag

团队网站

☐ 将“团队网站”设置为“我的主页”中的默认页

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Web

纹理文件夹

%PUBLICDOCUMENTS%\Autodesk\Inventor %RELEASE%\Tex

选项

打开文件...

文件命名默认值...

文件命名默认值

结构件生成器 三维布管 镜像和复制

显示名称

文件名

☐ 使用文件名作为显示名称

结构件部件: Frame 19283746

<文件名>

BOM 表结构 虚拟件

骨架零件: Reference Skeleton

Reference Skeleton

端盖: [Display Name]

<资源中心显示名称>

结构件: AISC C 3 x 3 - 100

<标准> <库存编号> - <长度>

单位小数标记

文件名: Bolt M3.5x20. ipt

默认值

由操作系统定义的小数标记: . (句号)

Frame 19283746. iam

Frame <唯一编号>

位置: [部件路径]\部件\Frame

<部件路径><部件名称>\Frame

Skeleton 19283746. ipt

Skeleton <唯一编号>

Plate Fillet Corners (in) 19283746. ipt

Plate <族名称> <唯一编号>

AISC C 3 x 3 19283746. ipt

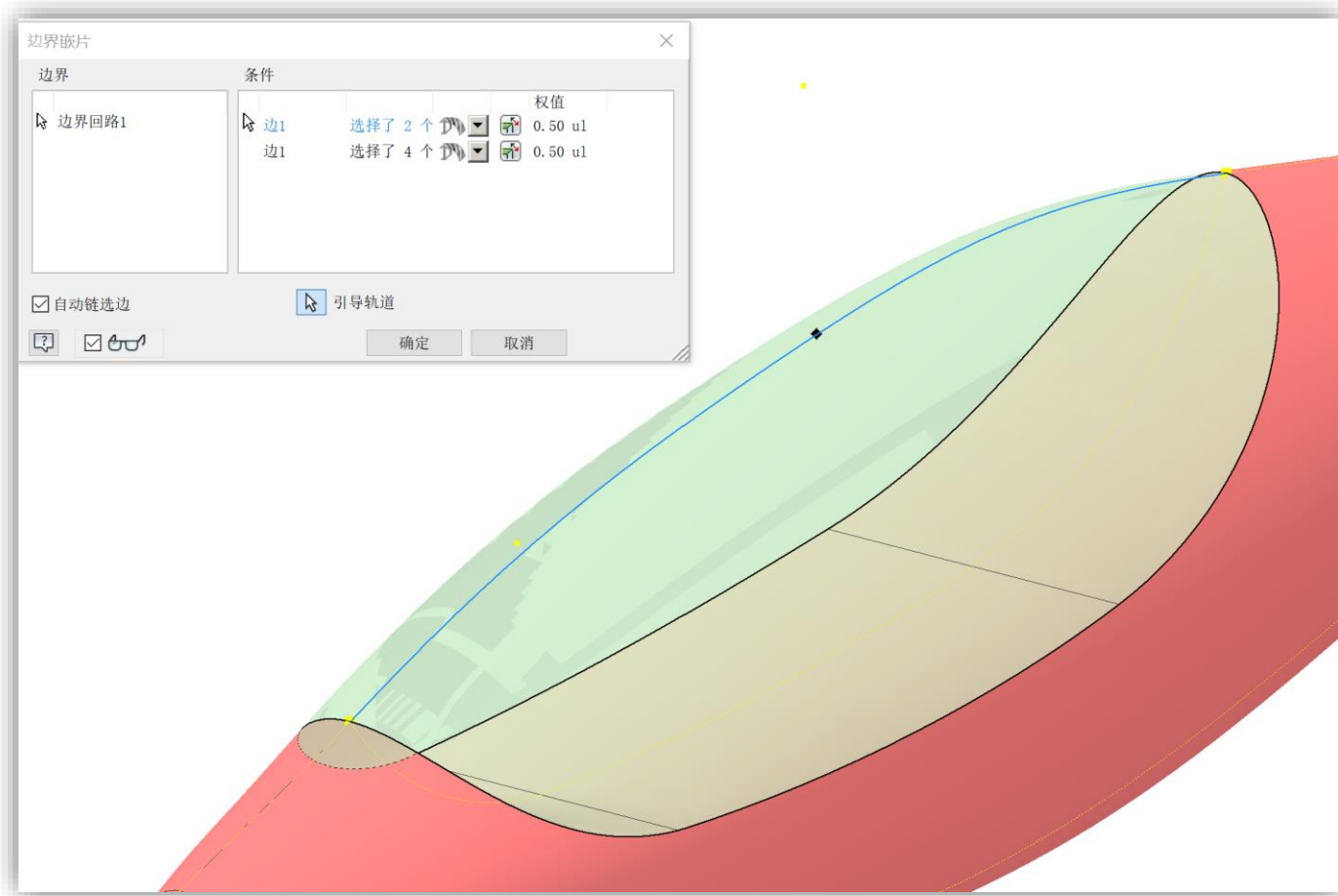
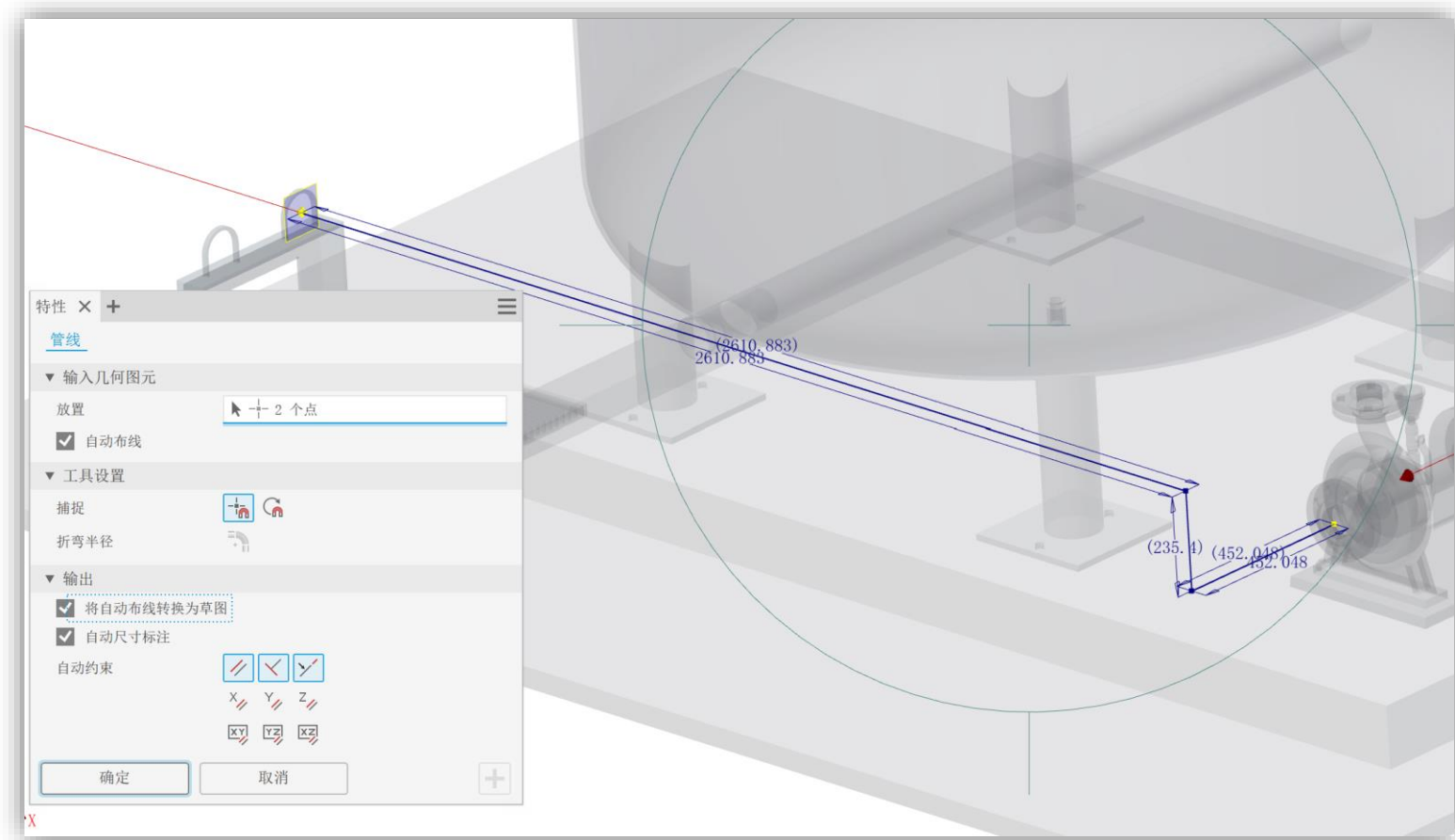
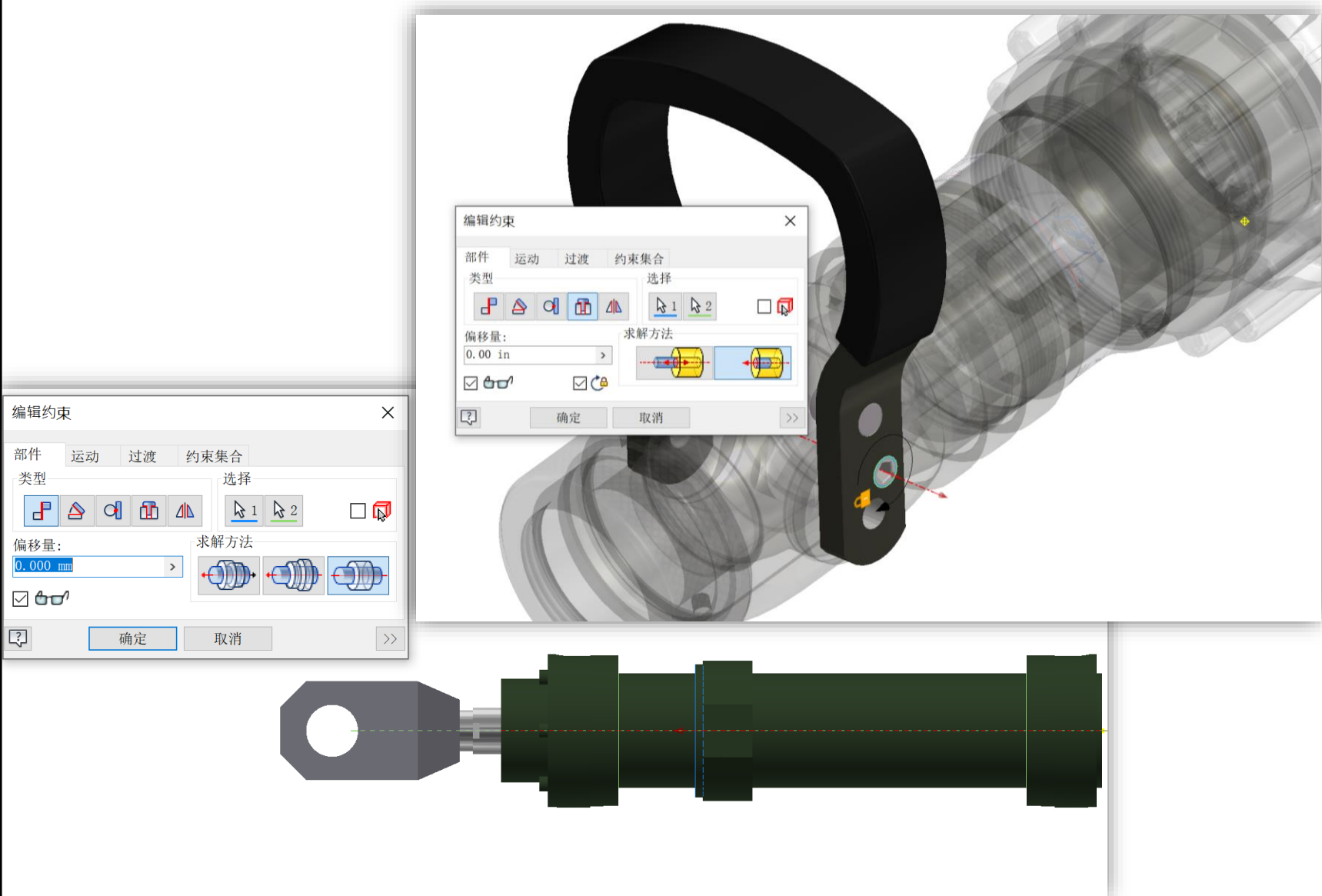
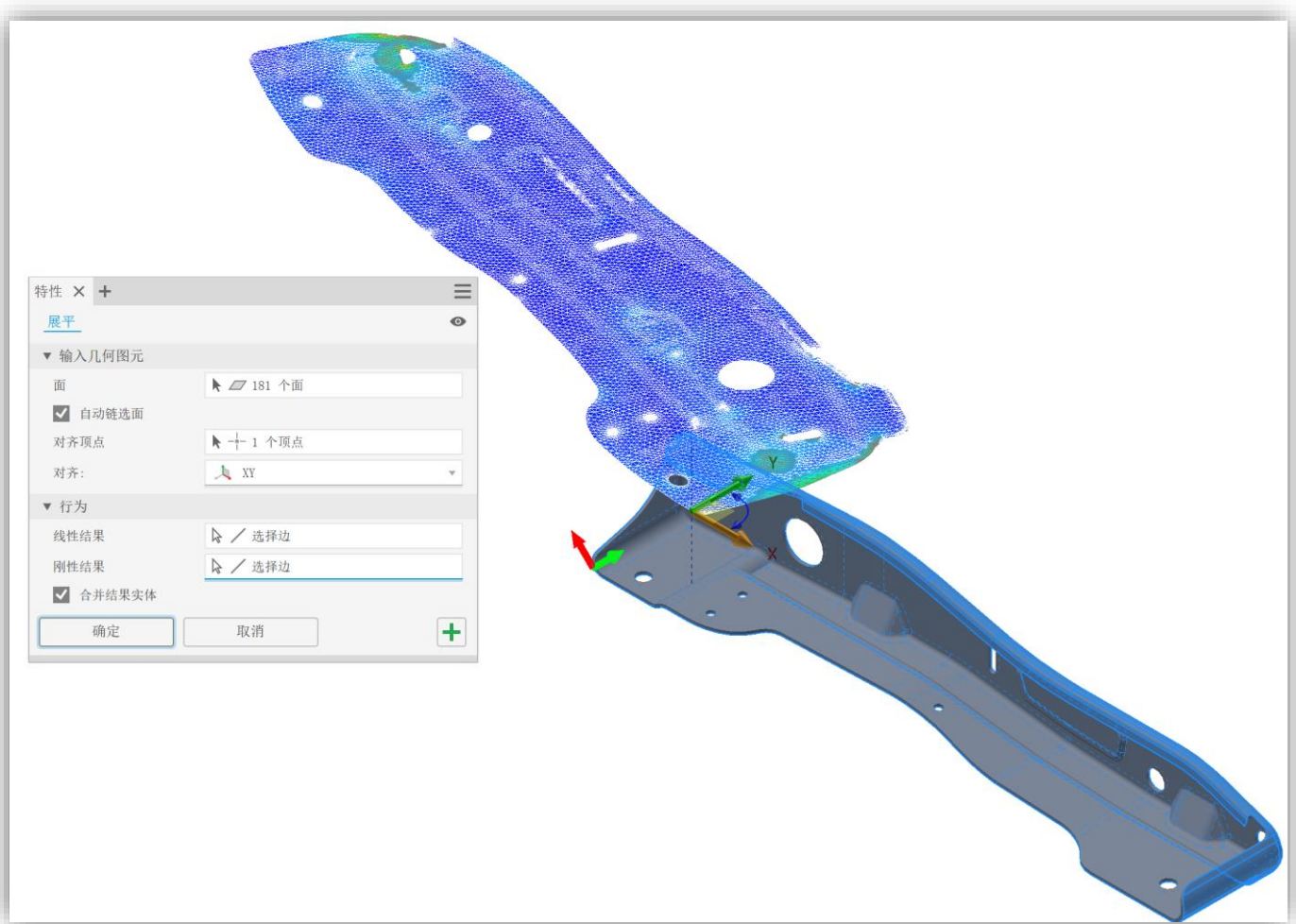
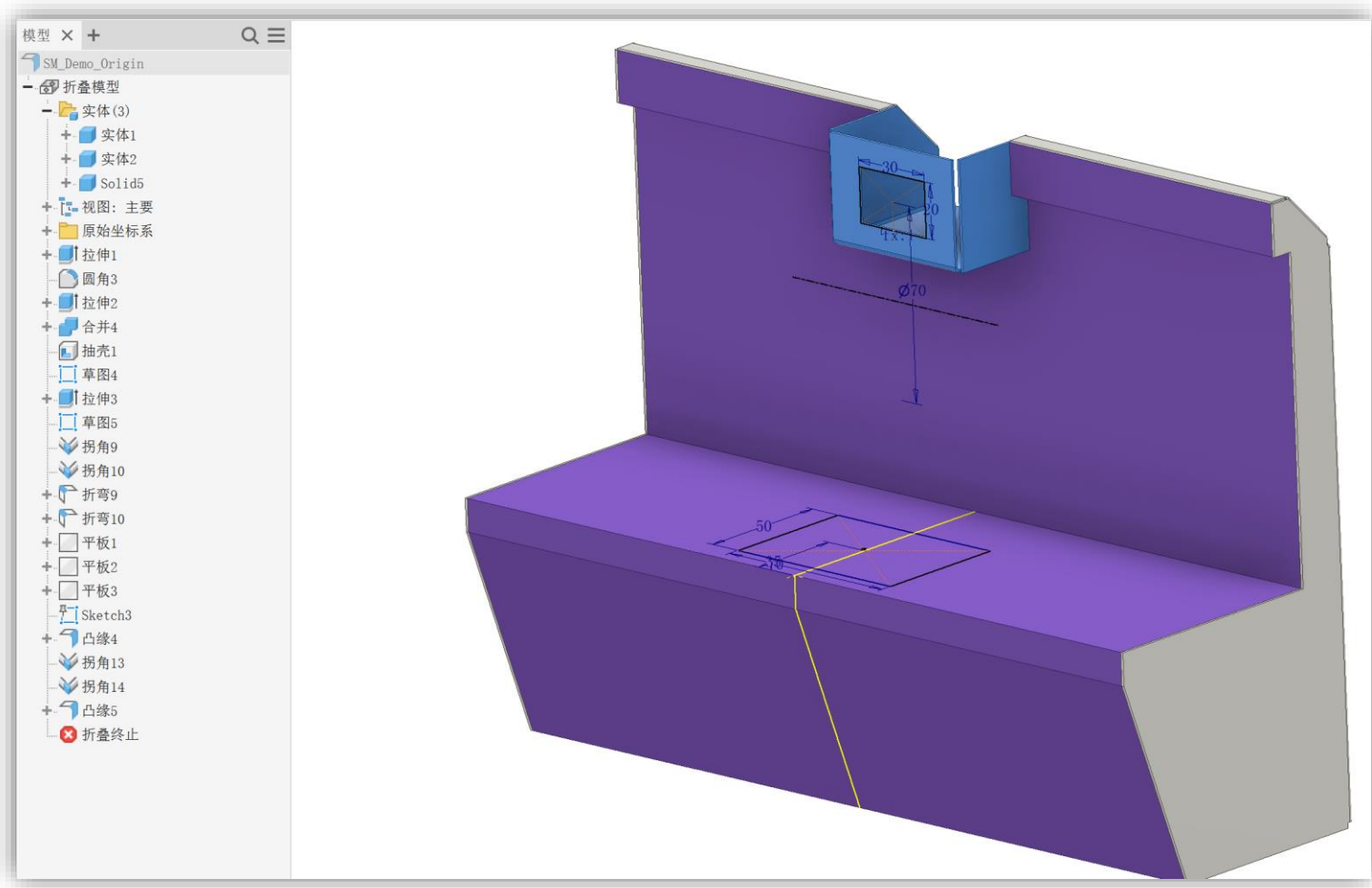
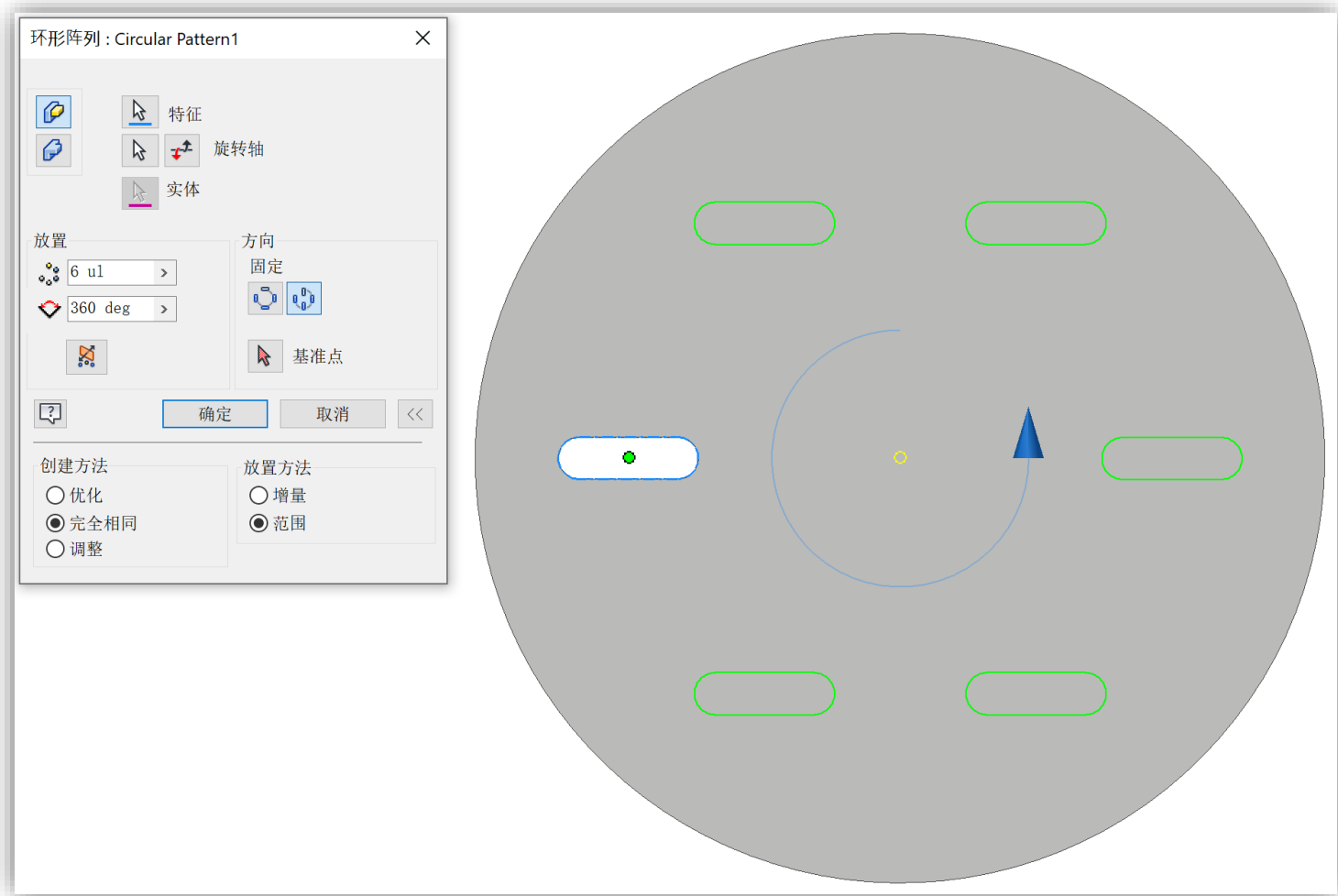
<标准> <库存编号> <唯一编号>

☐ 由于大小更改而覆盖当前文件

☐ 使用部件单位

OK Cancel

增强内容涉及Inventor的各个方面



资源

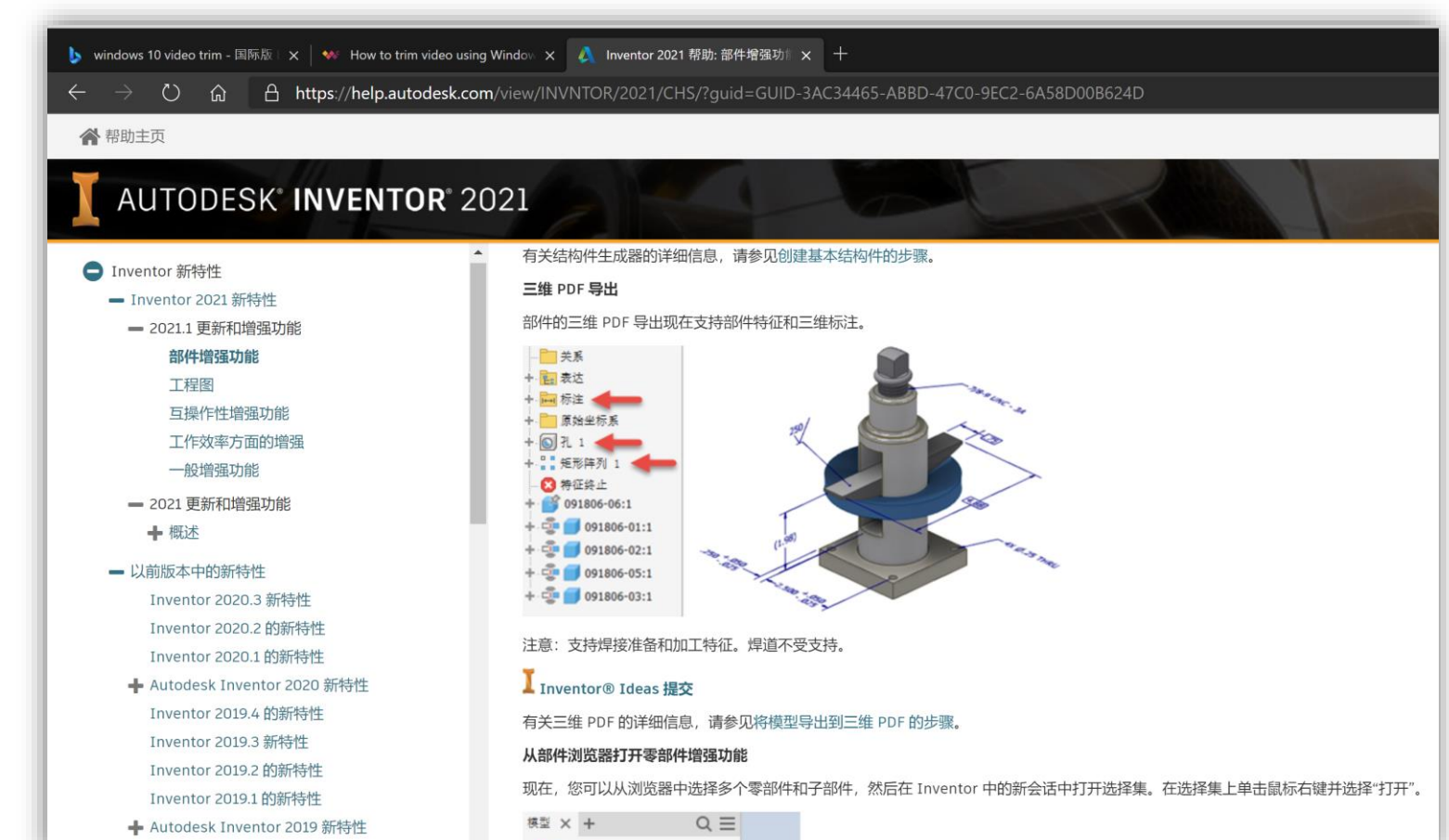
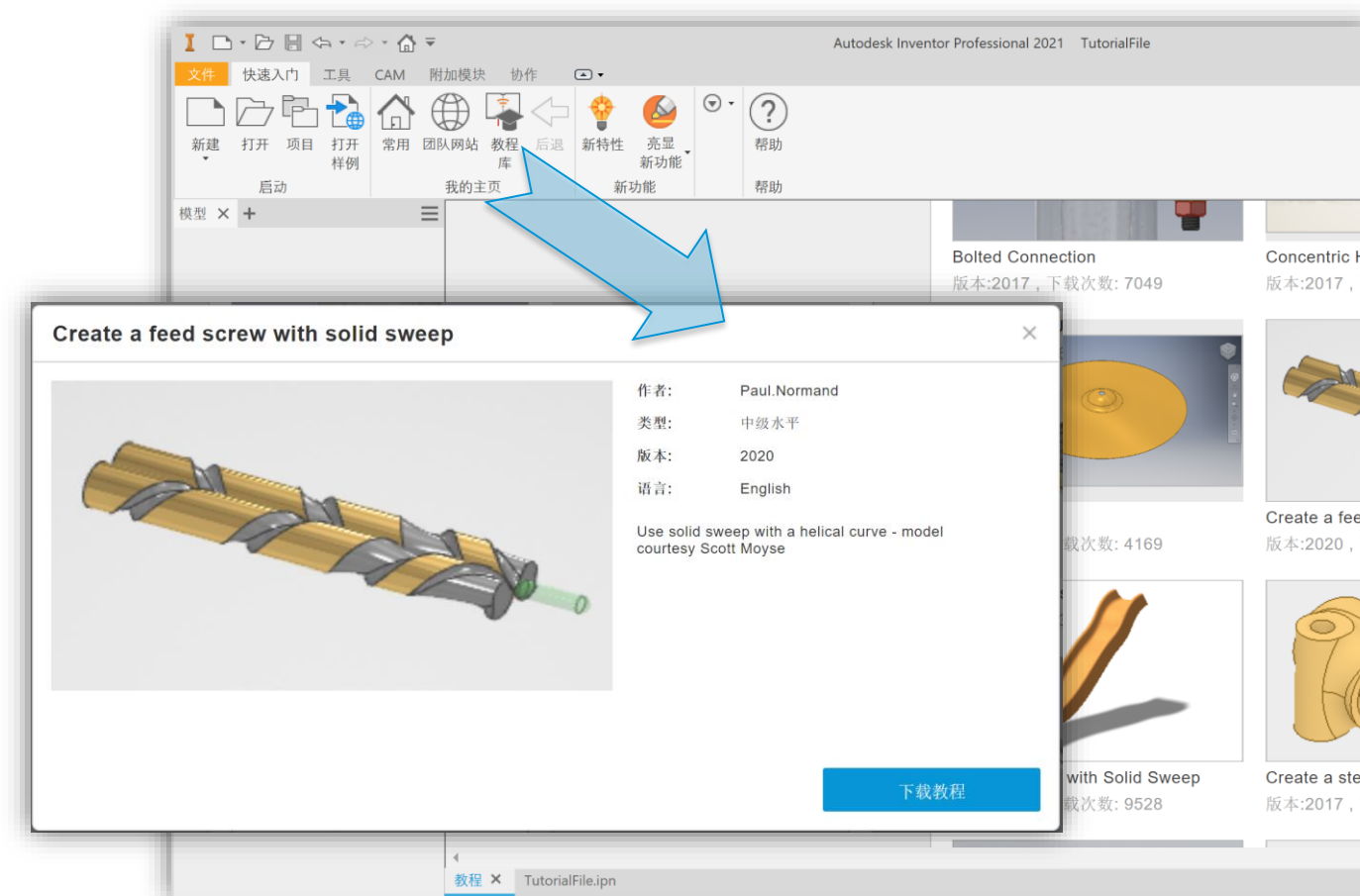
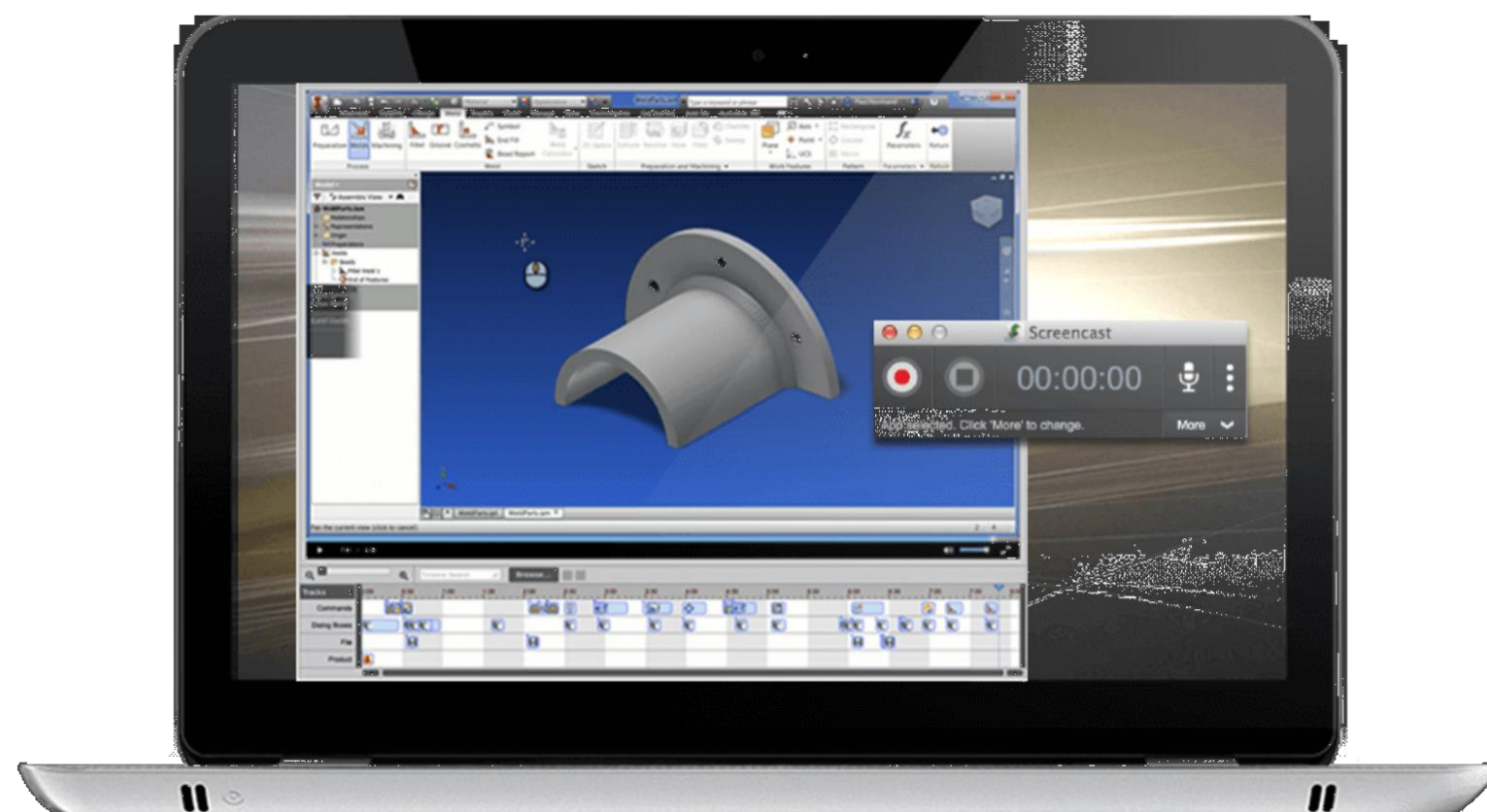


- Autodesk 录屏软件-Screencast ([链接](#)) ;
- Inventor 教程库 (Guided Tutorial)
- Inventor Help (在线帮助文档, 新功能介绍)

- Inventor 产品技术应用讨论区 ([链接](#))
- Inventor 反馈社区 ([受邀参加](#))
- 欧特克大视界 ([微信ID: autodeskvision](#))

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- Inventor Forums 论坛
 - Inventor Ideas 用户金点子
- <https://forums.Autodesk.com>

-
- 邮件: mfg.usergroup.china@autodesk.com



Screenecast 链接

- 指定尺寸对齐方向;
- 草图驱动阵列
- 引出件号沿指定边对齐;
- 参考件在工程图的多种显示方式
- 进给螺杆例子
- 实体扫掠



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