

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

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讲师介绍

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刘静宣，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产品研发中心，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

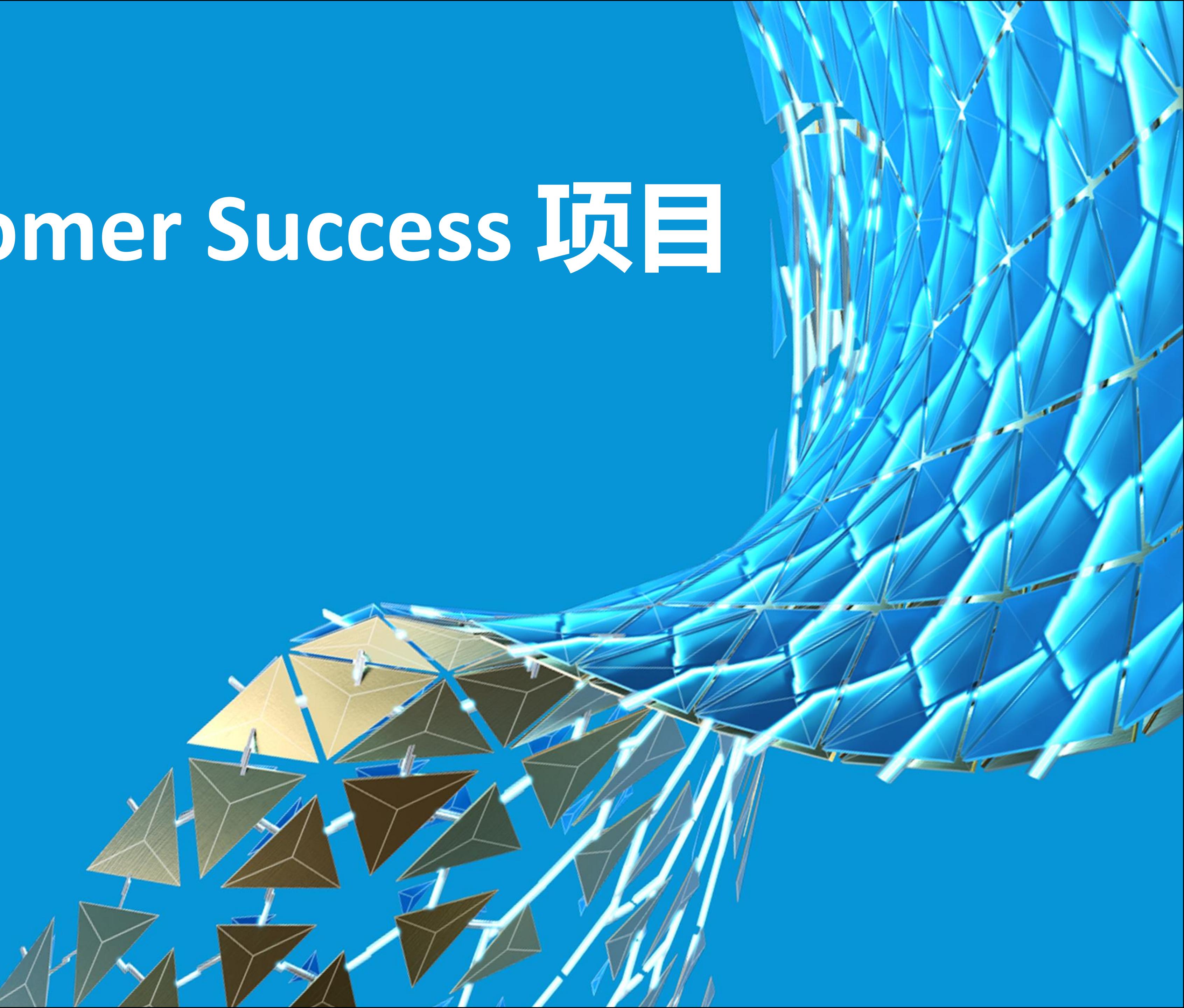
目的

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

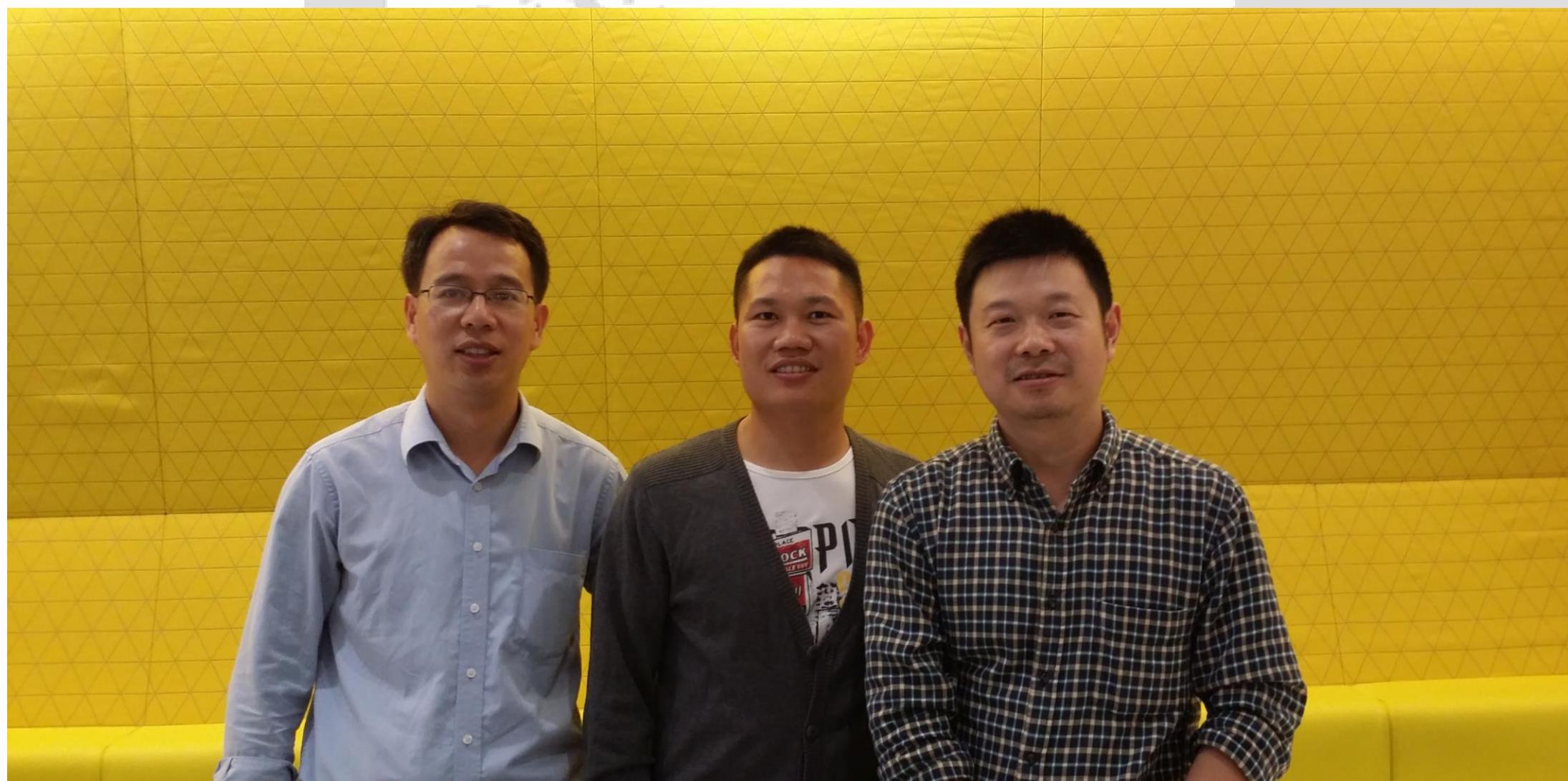
Inventor Customer Success 项目

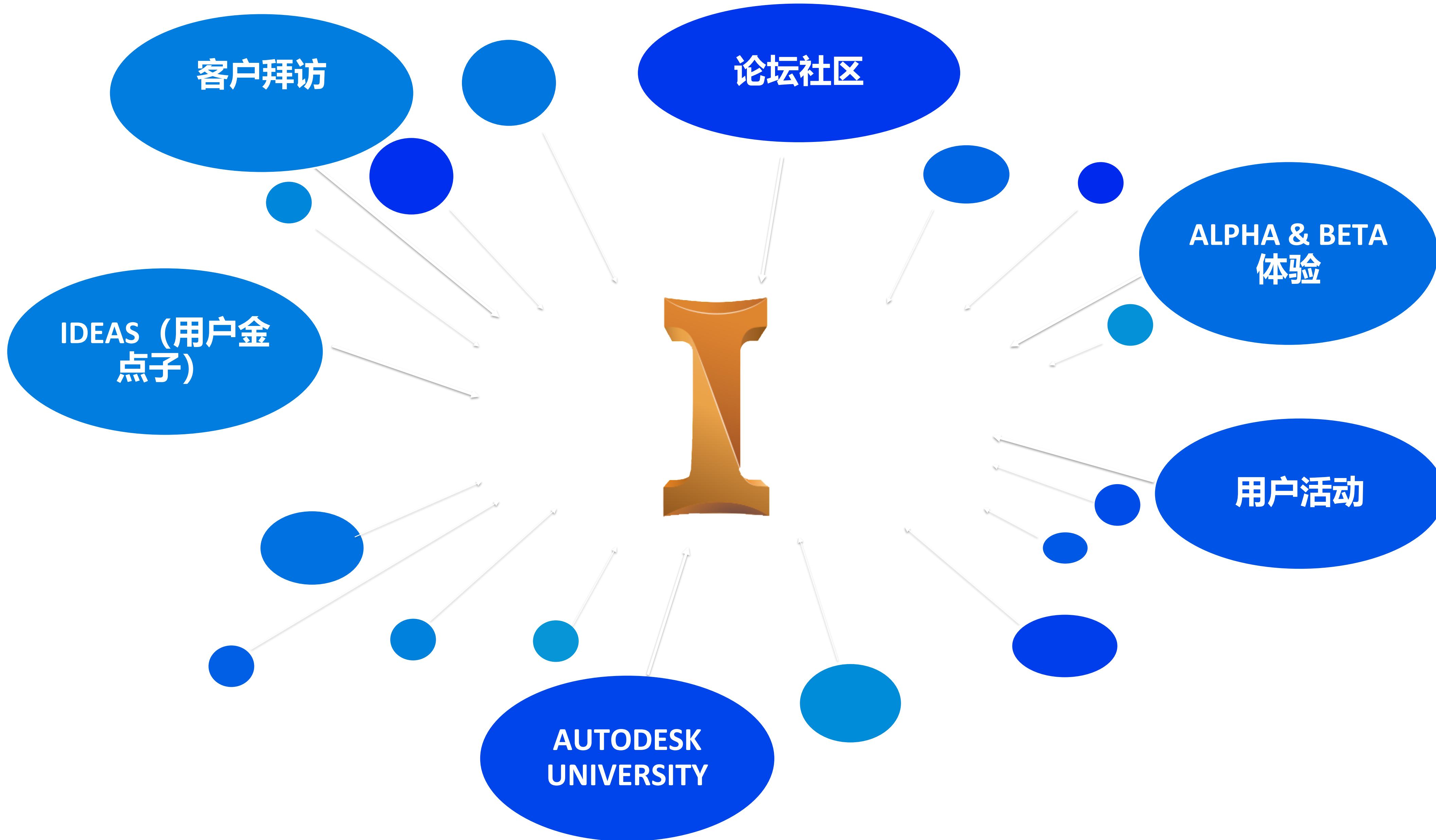


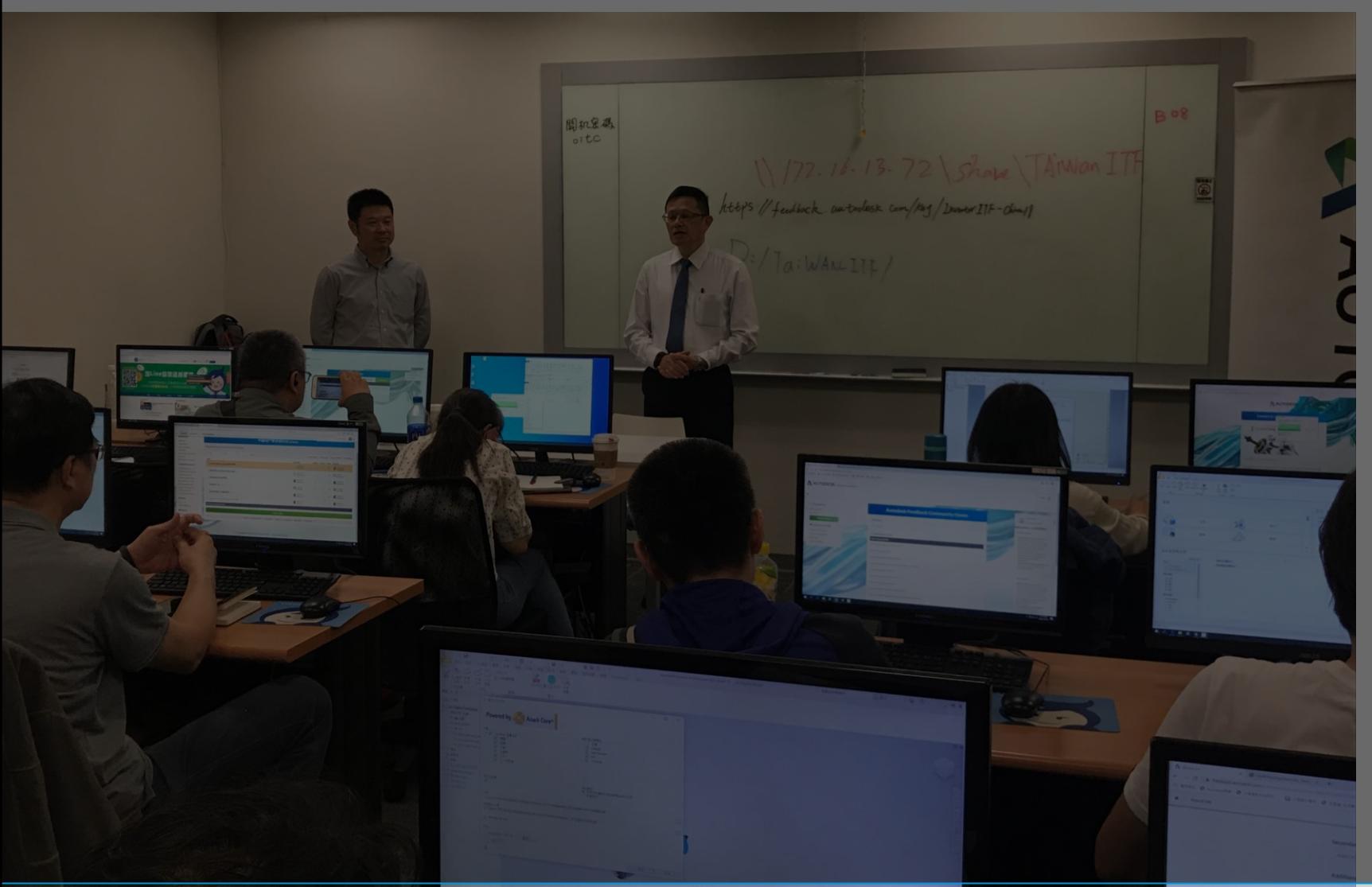
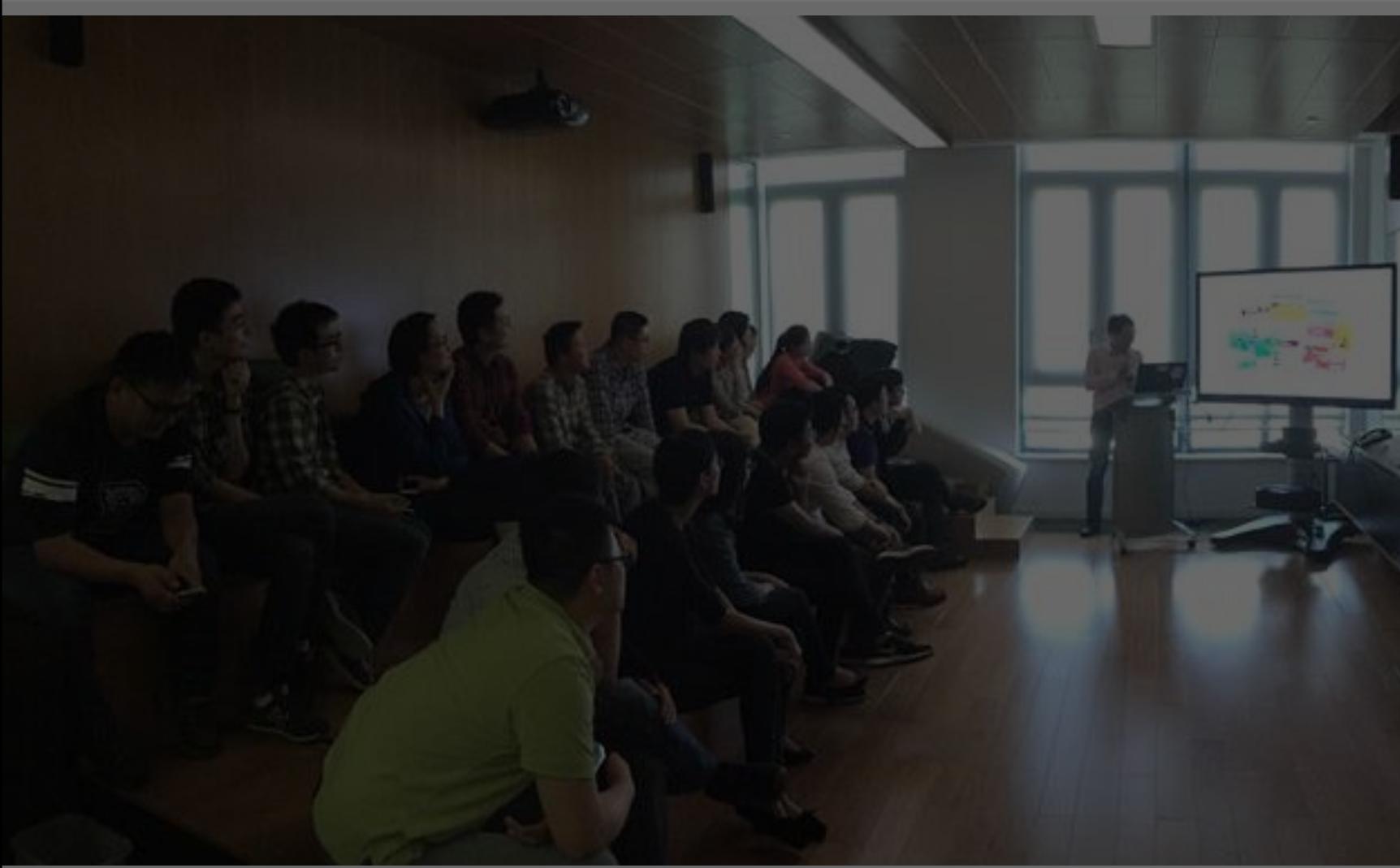
Inventor Customer Success项目

- 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

The image is a composite screenshot illustrating various communication and collaboration tools:

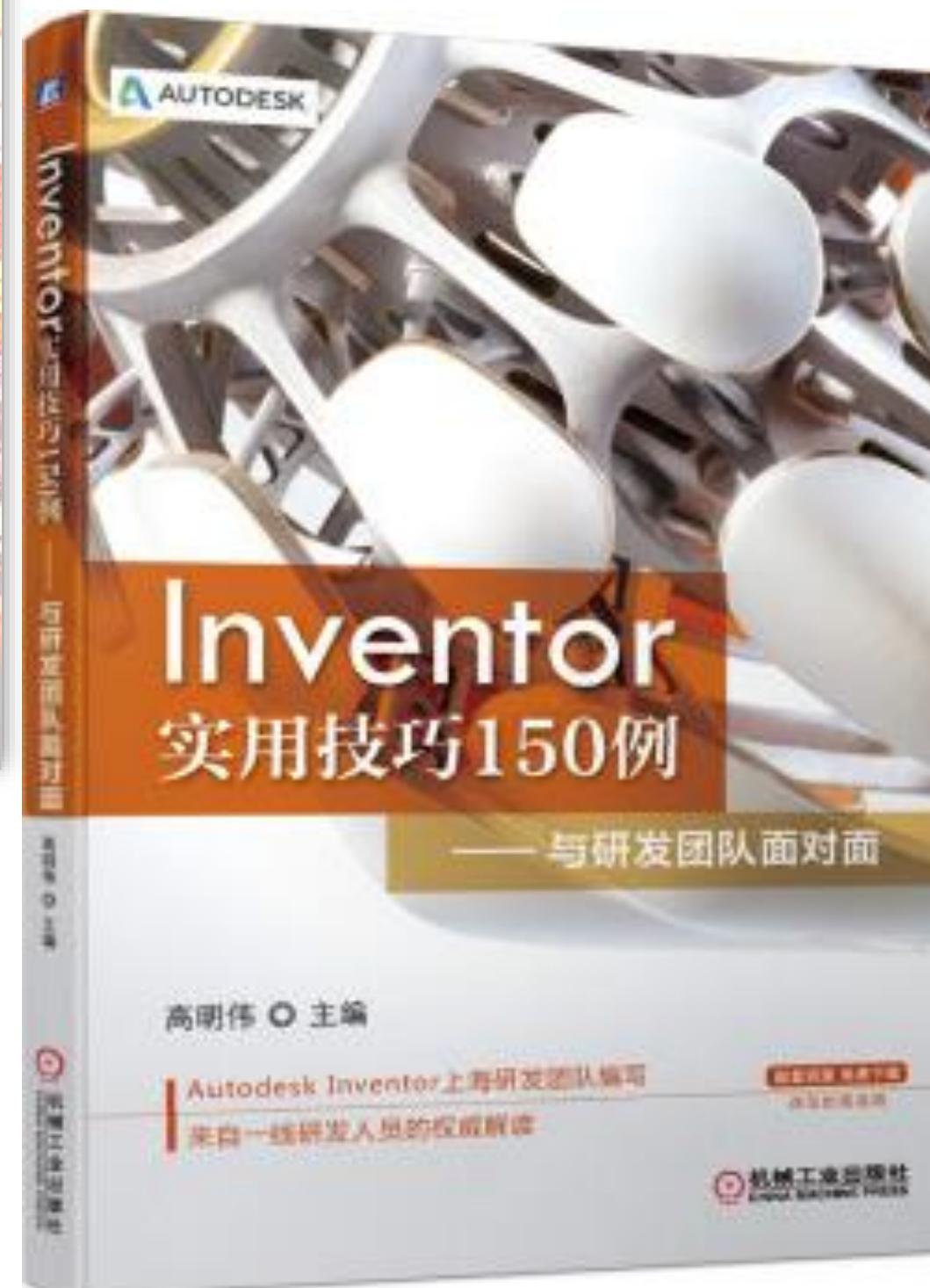
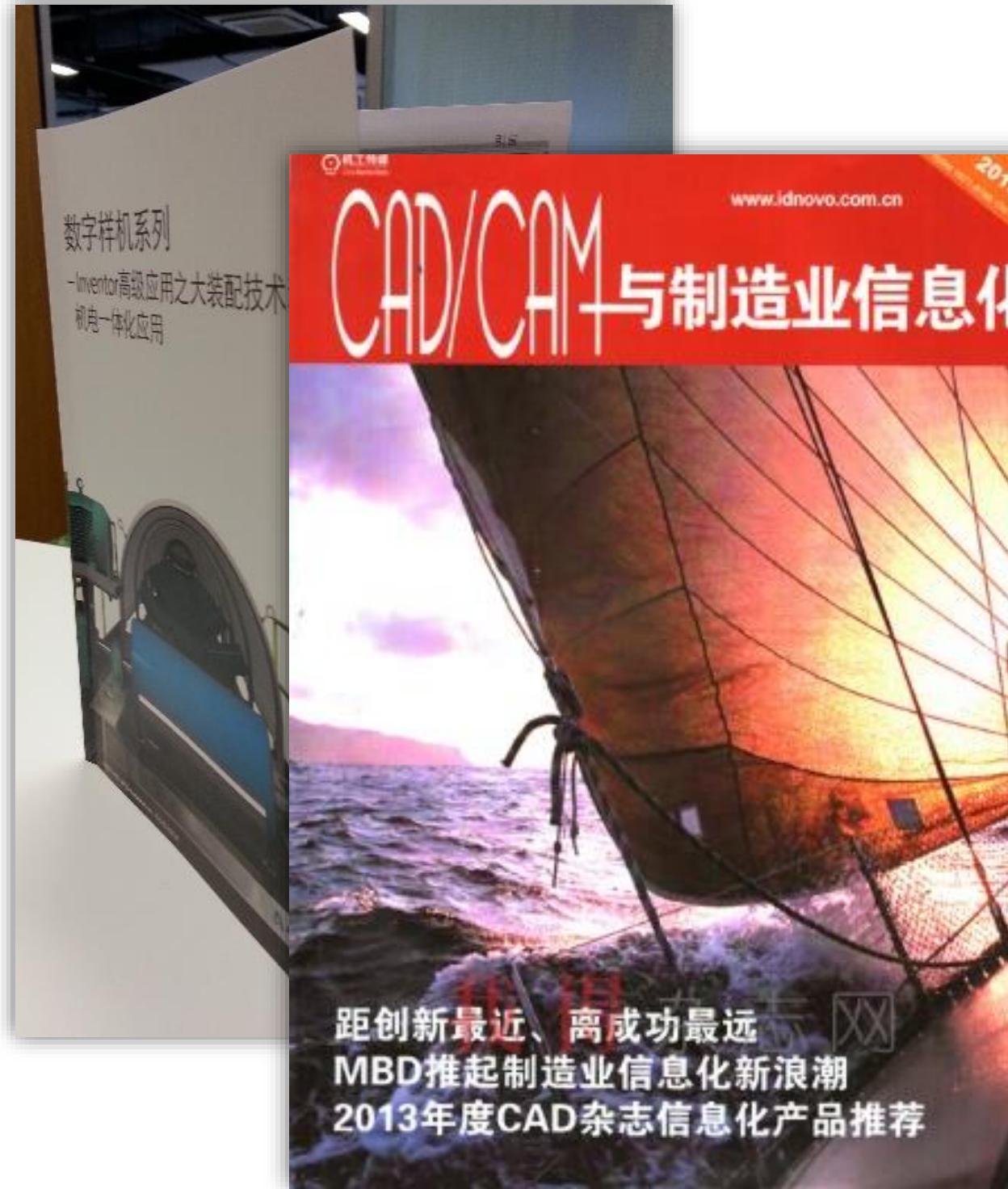
- Top Left:** A Zoom meeting interface showing participants' video feeds and a participant list.
- Middle Left:** A screenshot of the Autodesk Inventor software interface, specifically the "Part" tab, with a list of hidden documents.
- Bottom Left:** A list of features related to document management in the software.
- Center:** A large title "多显示器支持" (Multi-Monitor Support) overlaid on the Inventor interface.
- Right Side:** A Mural board titled "Inventor现代化体验 问与答" (Inventor Modern Experience Questions and Answers). It contains several user comments and responses, some highlighted with yellow boxes, and a screenshot of the Inventor ribbon bar.
- Bottom Right:** A title "特性面板UI优化" (Property Manager UI Optimization) with a row of small circular icons.

Document Management Features (Bottom Left):

- Hide my home and other documents from arrangements
从排列中隐藏主页和其他文档
- Show hidden documents in the dropdown menu
在下拉菜单中显示隐藏的文档
- Drag documents to an empty document frame
拖动文档到一个新的窗口
- Quickly restore all hidden documents
快速恢复所有隐藏的文档
- Auto restore documents when selected manually
手动选择后将自动恢复文档
- When arranged hidden documents are placed in the bottom left frame
排列时，隐藏的文档将放在左下窗口中

Inventor Modern Experience Questions and Answers (Right Side):

- 我现在还是试用超宽屏幕，多显示器支持对我有什么价值？
我们提供的改进也能在单个显示器中发挥作用，包括快速拖放窗口等功能。
- 目前我可以拖拽文件标签来拖动窗口，但是不能拖动边框，会改进吗？
我们内部已经明确了这个需求，正在解决一些技术问题，请持续关注。
- 能否隐藏一个或几个后窗口，立刻让其余几个窗口撑满屏幕，不需要手动去排列。（隐藏一个或几个后窗口后留出空间好别扭）
对，把最小化按钮加回后单击，还能看到有最小化的图标。
- 是不是就增加了一个最小化按钮？
用标签背景颜色区分，很好找到自己的文档。
- 您指的是不想显示右上角的帮助按钮吗？
可以重新把自定义工具栏放到右侧。
- 是不是修改了颜色之后，文档的名字地方能修改颜色么？可以改变颜色么，文档的名字的地方不是，需要修改文件那样，可以随着修改颜色，可以很好的区别对背景颜色很好找到自己的文档。
对，把最小化按钮加回后单击，还能看到有最小化的图标。



← | Inventor用户俱乐部 |

iLogic在转炉倾转过程的应用 (二) -欧特克 (中国) 软件研发有限公司 刘静宜

Inventor 2021 的新功能和公开线路图更新
欧特克大视界 4/5

Inventor 2021 的新功能

Inventor 团队来说，3月底总是一年中最激动人心的。为我们准备好为用户提供新版本的 Inventor。31 日开始，Inventor 用户将开始在您的 Autodesk 帐户和 Autodesk Desktop App 中看到所有的 Inventor 2021。Inventor 2021 是过去 12 个反馈、市场研究和开发工作的结晶，而且我们认真真正想要的结果。下面是 Inventor 2021 中提更新和增强功能的概述。

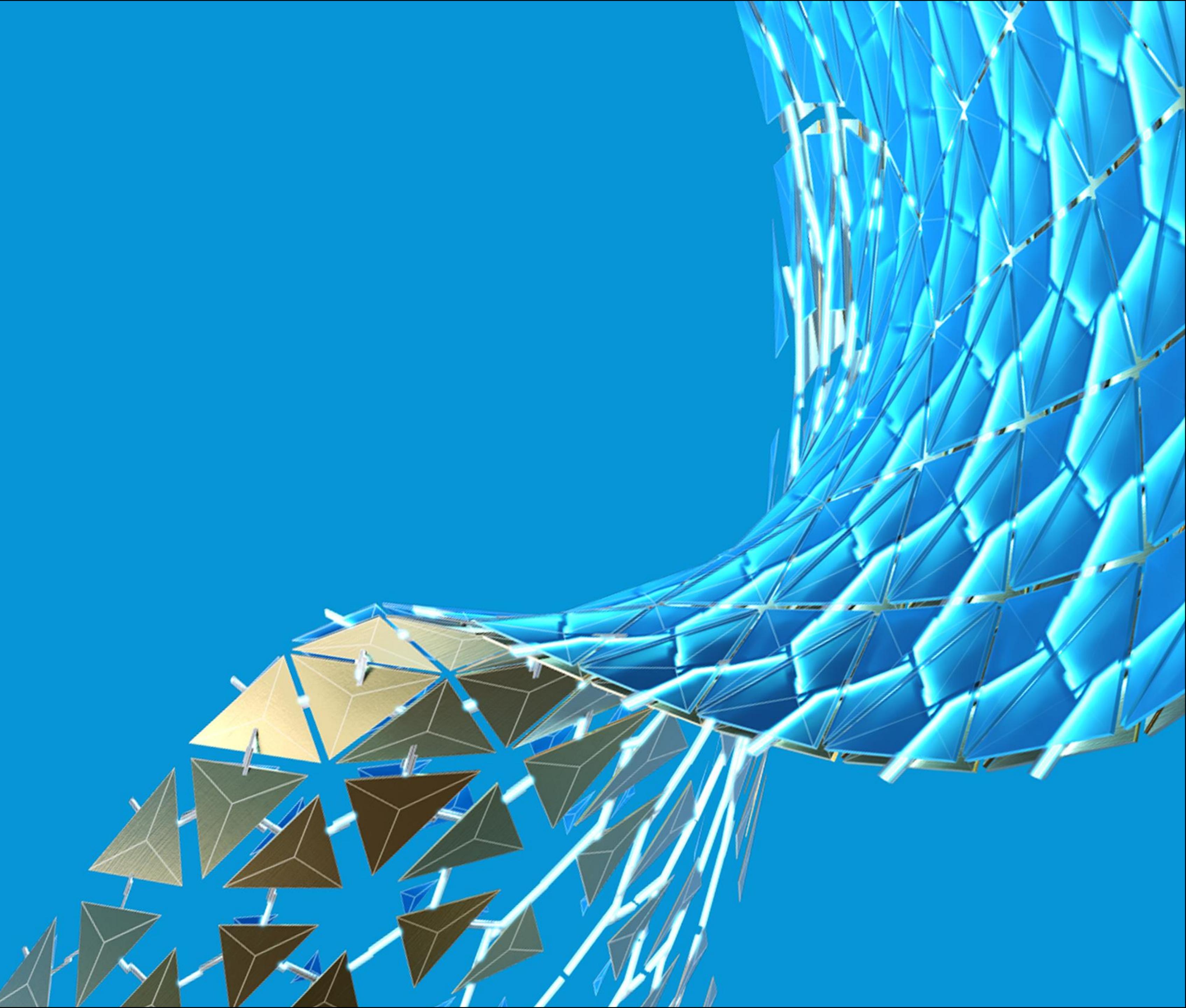
体验

12 iLogic 二 Autodesk A360

最新活动 干货分享 未来智造

微信ID： autodeskvision

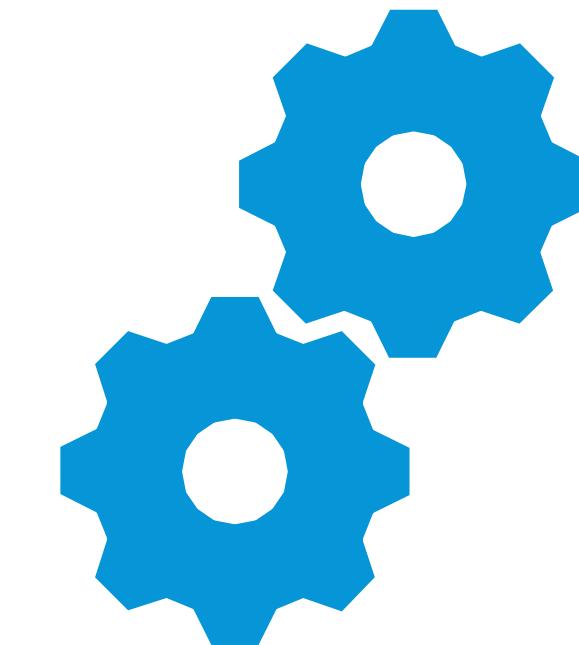
用户互动



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

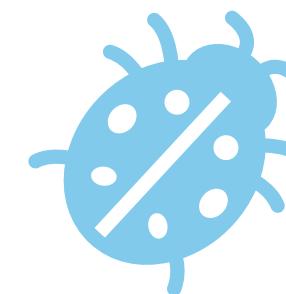
- 理解用户的问题，理清工作流程和影响面；
- 设计验证和功能验证；
- 了解行业的趋势和发展



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）

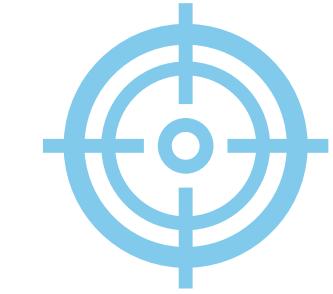


关键点：

- 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：

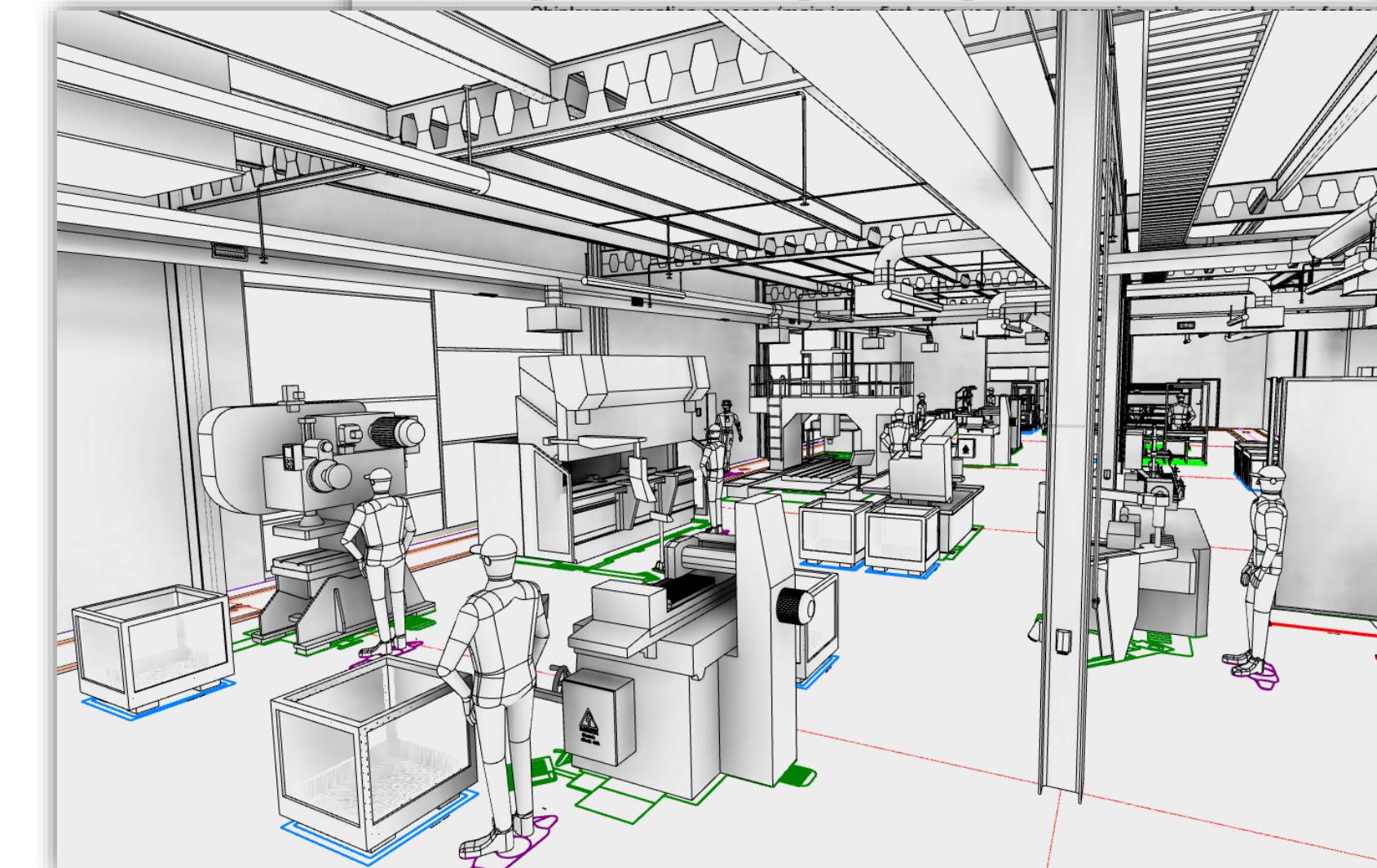
- 需求的普遍性；
- 对用户的影响，行业价值；
- 是否有技术的依赖性；
-

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

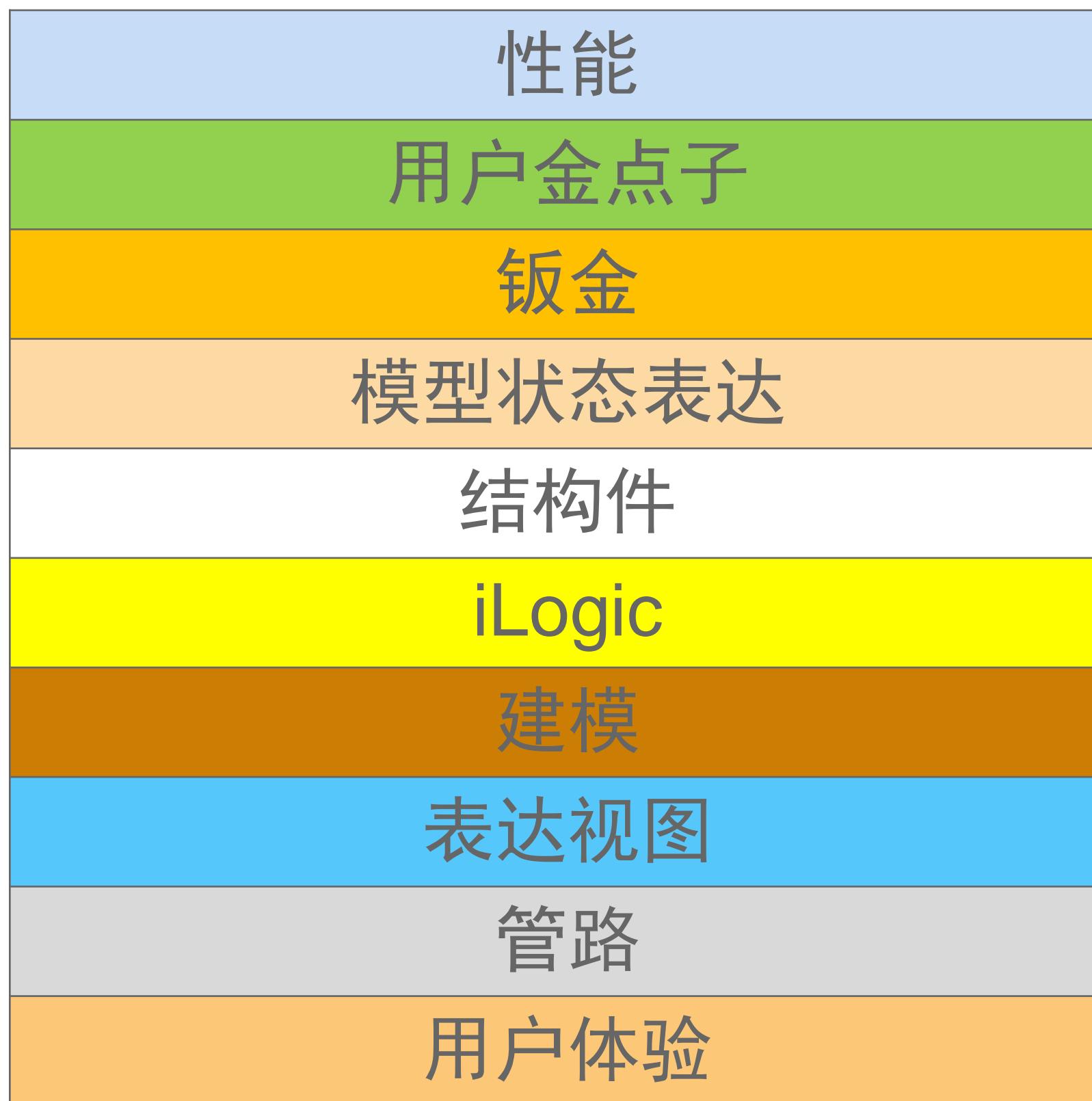
- Full LoD model with anyCAD referenced CATIA assemblies Inventor 2021
(\VAULT_Arbeitsordner\CAD\TH02\VP_BR46x\MaE\SI\H02_MOPF_SI_Vorrichtungen_FDU_H.iam):
 1. Open model without Vault-Add-In 4:15 minutes (load full 7:30 minutes)
 2. Executing update process very time consuming
 3. Change LoD from <shrink>LOD 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\TH02\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\TH02\VP_BR46x\MaE\Dat_In\EBZ\20190405\SI\TH02DSI11110WZ01000_V00K_ZB_STATION_1110_RP.iam):
 - Shinkwrap 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\TH02\VP_GMOPF\MaE\Dat_In\EBZ\20190405\SI\TH02DSI11110WZ01000_V00K_ZB_STATION_1110_RP.iam):
 - Shinkwrap 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



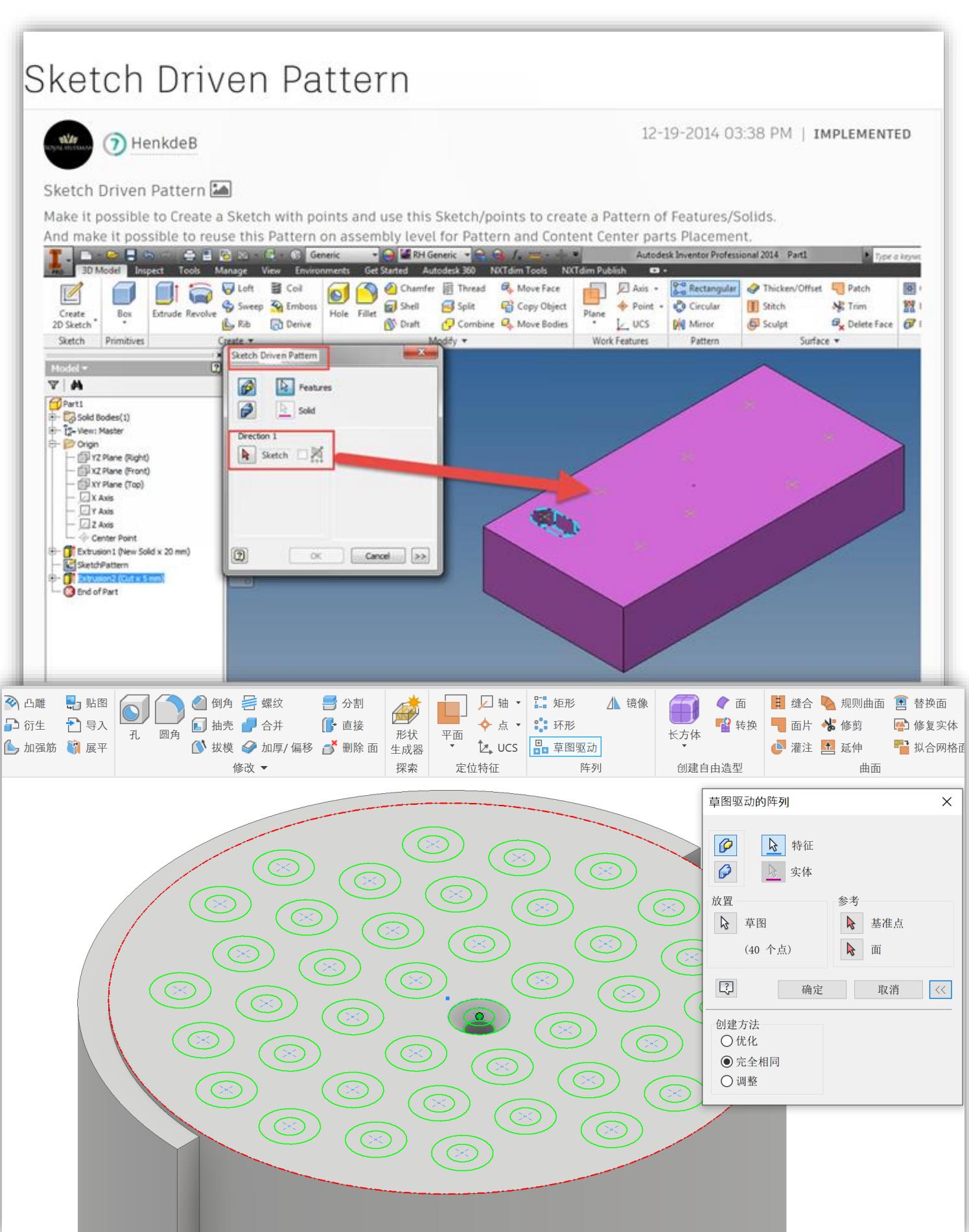
用户需求



■ 工程图 ■ 装配 ■ 零件 ■ 用户界面 ■ 工程图 ■ 其它



草图



Project geometry as construction

02-21-2014 12:31 AM | IMPLEMENTED

Project geometry as construction

It would be good to have an option for projected geometry to be construction lines. This would eliminate having multiple closed profiles in a sketch when the autoproject edges option is enabled; yet allow reference to other features / parts.

应用程序选项

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

启用平视显示仪 (HUD)

设置...

捕捉到网格

在创建曲线过程中自动投影边

自动投影边以创建和编辑草图

新建草图后，自动投影零件原点

将对象投影为构造几何图元

创建和编辑草图时，将观察方向固定为草图平面

在零件环境中

在部件环境中

点对齐

默认情况下在插入图像过程中启用“链接”选项

根据初始尺寸自动缩放草图几何图元

三维草图

新建三维直线时自动折弯

草图显示

透过着色模型显示的草图的不透明度 60 %

Add CTRL behavior to Horizontal and Vertical sketch constraint.

4 05:28 PM | ACCEPTED

Add CTRL behavior to Horizontal and Vertical sketch constraint.

If you choose a horizontal constraint, select the horizontal axis.

自定义

功能区 键盘 标记菜单

类别 (C) 草图 全部

键	命令名	类型	类别
T	控制顶点样条曲线	草图	
	插入 AutoCAD 文件	草图	
A	文本	草图	
C	旋转	草图	
	显示约束	草图	
S	构造	草图	
Z	样条曲线	草图	
E	椭圆	草图	
H	水平	快捷方式	草图
L	十点, 中心点	草图	
	环形阵列	草图	
M	由三点圆弧确定的槽	草图	
	由中心到中心确定的槽	草图	
R	由圆心圆弧确定的槽	草图	
	由总长度确定的槽	草图	
I	直线	草图	
=	相切	草图	
	相切圆弧	草图	
D	矩形阵列	草图	
M	移动	草图	
J	垂直	Alias	草图
	等长	快捷方式	草图
	等长	快捷方式	草图
	精确输入	草图	

重设所有键

使用默认的多字符命令别名

描述

复制到剪贴板

实体扫掠

New Feature Sweep solid

OPTIONS ▾

Back to Idea Board < Previous | Next >

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

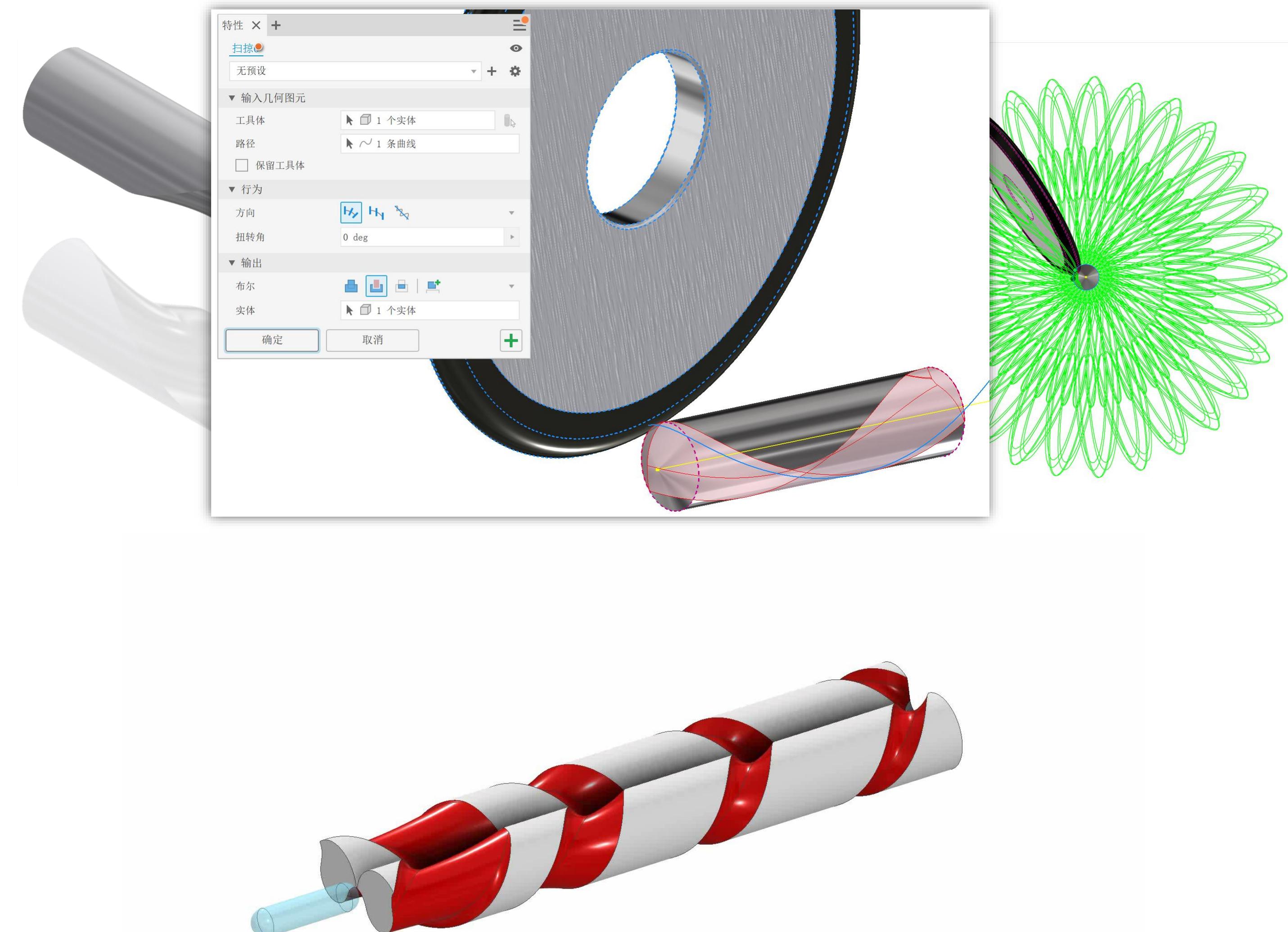
Report

Add tags

VOTE 609 Votes

63 Comments (13 New)

POST A COMMENT



干涉检查

Thread interference analysis filters

The screenshot shows the Autodesk Inventor Professional 2021 interface with a Bevel Gear assembly open. A callout box highlights the '干涉检查' (Interference Check) tool in the top menu bar. A modal dialog box titled '检测到干涉' (Detected Interference) is displayed, stating '共检测到 27 处干涉, 其体积总计为 1.963974E+03 mm^3.' (27 interference detected, total volume 1.963974E+03 mm³). The dialog lists the detected interference pairs and their volumes. The main workspace shows the 3D model of the bevel gear assembly, with several specific干涉 points highlighted by red circles.

OPTIONS ▾

m.colombo

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

18 C

干涉 激活接触识别器 公差 分析 测量 面域特性

干涉 检验 工具 CAM 管理 视图 环境 快速入门 附加模块 协作 机电

Autodesk Inventor Professional 2021 Bevel Gear_Demo

搜索帮助和命令...

jingyi.liu@aut...

干涉

模型 | 造型

Bevel Gear_Demo.iam

关系

表达

视图: Default

位置

详细等级 : 主要

所有零部件均已抑制

所有零件均已抑制

资源中心零件均已抑制

原始坐标系

GEAR2P

GEAR2W

BG-P-001:1

GEAR1P

GEAR1W

BG-P-004:1

DIN 720 30207 - 35 x 72 x 18.25:1

DIN 625 - T1 6203 - 17 x 40 x 12:1

DIN 625 - T1 6201 - 12 x 32 x 10:1

DIN 617 RNA RNA4903 - 22 x 30 x 13:1

DIN 472 40x1.75:1

DIN 472 30x1.2:1

DIN 472 30x1.2:2

DIN 6885 - A 5 x 5 x 14:1

DIN 6885 - A 5 x 5 x 14:2

SHAFT3_MM:1

DIN 6888 (I) 6 x 7.5:1

BG-P-003:2

DIN 720 30207 - 35 x 72 x 18.25:2

零部件阵列 2:1

元素:1

ISO 4762 M5 x 12:1

元素:2

ISO 4762 M5 x 12:2

元素:3

元素:4

元素:5

元素:6

元素:7

检测到干涉

重置 确定

其他信息:

项	Part 1	Part 2	体积
I...	BG-P-001:1	GEAR1P	17.73
I...	GEAR2P	GEAR2W	0.502
I...	GEAR1P	GEAR1W	0.509
I...	GEAR2P	DIN 625 - T1 6203 - 1...	2.626
I...	GEAR1P	DIN 625 - T1 6201 - 1...	89.64
I...	GEAR1W	DIN 6885 - A 5 x 5 x ...	8.575
I...	DIN 720 30207 - 35 x ...	SHAFT3_MM:1	3.172

干涉类型

常规 (3)

螺纹 (20)

非螺纹资源中心零件 (4)

Bevel Gear.iam Bevel Gear_Demo.iam

按 F1 获得帮助

53 27

工程图

Smarter Diameter Dimensions

Back to Idea Board | < Previous | Next >

mratray 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

3/4-10 UNC - 2B
1.250
1.438

DETAIL A

ø11.61 **ø7.87** **ø3.94**
ø11.42

Custom alignment direction for Aligned Dimensions

10-06-2014 05:13 AM | IMPLEMENTED

brendan.henderson

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

11-11-2014 10:50 AM | IMPLEMENTED

exSWX

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 to this edge!!!

Align_balcon_BOM

Reference Parts and Shaded Drawing Views

04-30-2015 05:34 AM | IMPLEMENTED

mthomas

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.

No Reference Parts
Reference Part

工程视图

零部件 模型状态 显示选项 恢复选项 成员 隐藏线计算 显示样式 分别参考数据 (●) 所有实体 (○) 隐蔽线 (□) 捕捉设置 (○) 参考数据 (○) 隐蔽线 (□) 尺寸类型(I) (●) 捕捉设置 (○) 参考数据 (○) 隐蔽线 (□) 对齐(A) (●) 缩放(Z) (○) 平移(P) (○) 上一视图 (○) 如何(H)... (●) 对齐(A) (●) 缩放(Z) (●) 平移(P) (●) 上一视图 (●) 帮助主题(H)... (●) 重复 自定义用户界面(R) (●) 选择其他(Q)... Ctrl+A (●) 删除(D) (●) 编辑引出序号(D)... (●) 水平(H) (●) 竖直(V) (●) 对齐(A) (●) 缩放(Z) (●) 平移(P) (●) 上一视图 (●) 帮助主题(H)... (●) 水平偏移(E) (●) 竖直偏移(O) (●) 水平偏移(O) (●) 至边(T) (●) 模型焊接符号 (●) 模型标注

ASKA

透明设置

Sketches should always be visible through objects when selected.

OPTIONS ▾

Back to Idea Board | Previous | Next >

DanSolo_ 01-18-2015 07:38 AM | IMPLEMENTED

Sketches should always be visible through objects when selected.

Sketches should **always be visible** through the sketch is selected or being created/edited. No exceptions!

There is no reason in 2015 that Autodesk should not have this issue resolved. When other software the user does not have to deal with having to section models or turn on wireframe mode in order to see what they are sketching.

Please fix this long over-due enhancement.

Inventor has some really nice features, but it has even more ridiculous quirks that make it...

应用程序选项

零件 iFeature 部件 资源中心

常规 保存 文件 颜色 显示 硬件 提示 工程图 记事本 草图

AutoCAD
 最小能量 - 默认张力

平视显示仪 启用平视显示仪 (HUD) **设置...**

捕捉到网格
 在创建曲线过程中自动投影边
 自动投影边以创建和编辑草图
 新建草图后，自动投影零件原点
 将对象投影为构造几何图元
创建和编辑草图时，将观察方向固定为草图平面
 在零件环境中
 在部件环境中
 点对齐
 默认情况下在插入图像过程中启用“链接”选项
 根据初始尺寸自动缩放草图几何图元

三维草图
 新建三维直线时自动折弯

草图显示
透过着色模型显示的草图的不透明度 **60** %

导入 (I)... 导出 (X)... 关闭 取消 应用 (A)

Transparent parts in assemblies

OPTIONS ▾

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cramayo 08-29-2013 02:25 AM | IMPLEMENTED

Transparent parts in assemblies

Users of SolidWorks know how easy it is to make a part transparent. This is real handy when you're checking hole patterns, clearances, or showing customers concepts. In Inventor you can change part transparencies by using "enable" or selecting an alternate material. If you uncheck enable, you lose the ability to select the part making it difficult to find in large assemblies. Personally, I don't like switching back and forth between materials to make a part temporarily transparent.

I would like Inventor to have an easy transparency option like this:

Transparent part option in drawing

OPTIONS ▾

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schnautza 05-23-2013 02:45 AM | IMPLEMENTED

Transparent part option in drawing

I'm working on an assembly that has some polycarbonate sheet as a cover. I'd like it if I could see what is behind the clear material in my drawing.

This is very different than wanting hidden lines - I DO NOT want to see every hidden line of the parts behind the transparent material. I only want to see the next visible part, maybe as a lighter line, but only the visible edges.

It would be great if there was an option when you right click a part to set as transparent in the drawing - similar to toggling visibility or hidden lines.

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.

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 type of issues:

Solution maybe something like a checkbox within The Notch feature.

Frame Generator - Notch With A Gap

OPTIONS ▾

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Easy end caps in frame generator

11-14-2012 09:37 PM | IMPLEMENTED

frame generator Add tags VOTE 289 Votes

pierre.masson

Easy end caps in frame generator

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

Add tags VOTE 88 Votes

Report 23 Comments (9 New) POST A COMMENT

特性 x + 插入结构件

无预设 放置 / 1 个实体

方向 旋转 -90.00 deg 绕选定点旋转 偏移 A 0.000 mm 偏移 B 0.000 mm 对齐 选择几何图元 确定 取消

特性 x + 增益

无预设 选择 位置 / 1 个面 行为 放置 轮廓 圆(英寸) 厚度 3.175 mm 偏移 - 轮廓 0.000 mm 材料 聚丙烯 外观 按材料 方向 旋转 0.00 deg 偏移 0.000 mm 确定 取消

特性 x + 斜接

上次使用 选择 结构件 / 4 个结构件 行为 类型 偏移方向 0.000 mm 确定 取消

特性 x + 槽

无预设 选择 结构件 / 1 个结构件 开槽刀具 结构件 / 1 个结构件 行为 槽轮廓 基本轮廓 延伸槽轮廓 垂直剖切 确定 取消

特性 x + 增益

无预设 选择 位置 / 1 个面 行为 放置 轮廓 1mm 偏移 - 轮廓 0.000 mm 材料 聚丙烯 外观 按材料 方向 旋转 0.00 deg 偏移 0.000 mm 确定 取消

特性 x + 插入结构件

无预设 放置 / 1 个实体

方向 旋转 90.00 deg 绕选定点旋转 偏移 A 0.000 mm 偏移 B 0.000 mm 对齐 选择几何图元 确定 取消

文件命名

Frame Generator Member Naming

01-10-2013 03:00 AM | IMPLEMENTED

cbenner

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 Content 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

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

jeff.perez

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

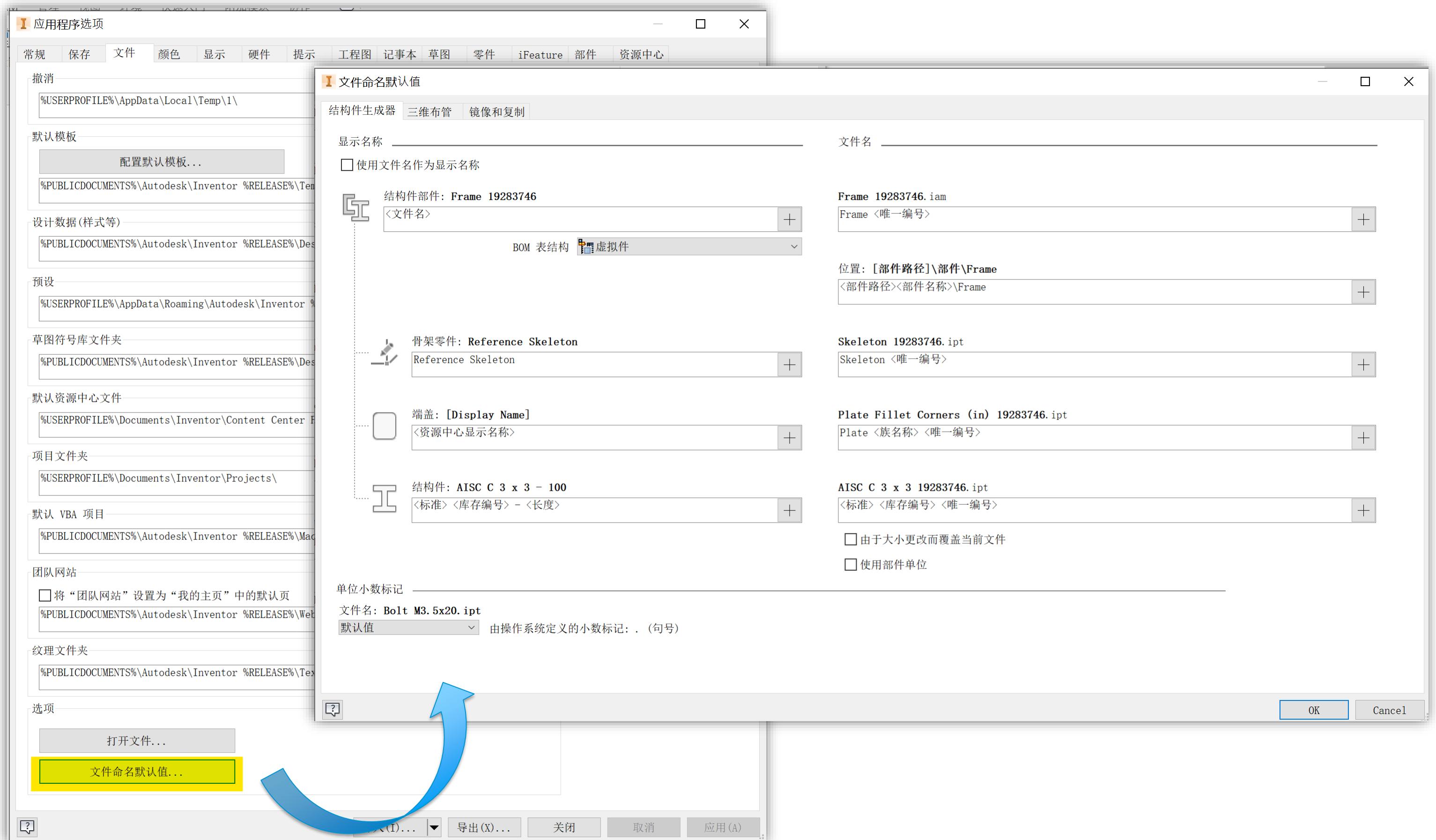
application options bolted connections, frame generator, Tube&Pipe

▲ Report

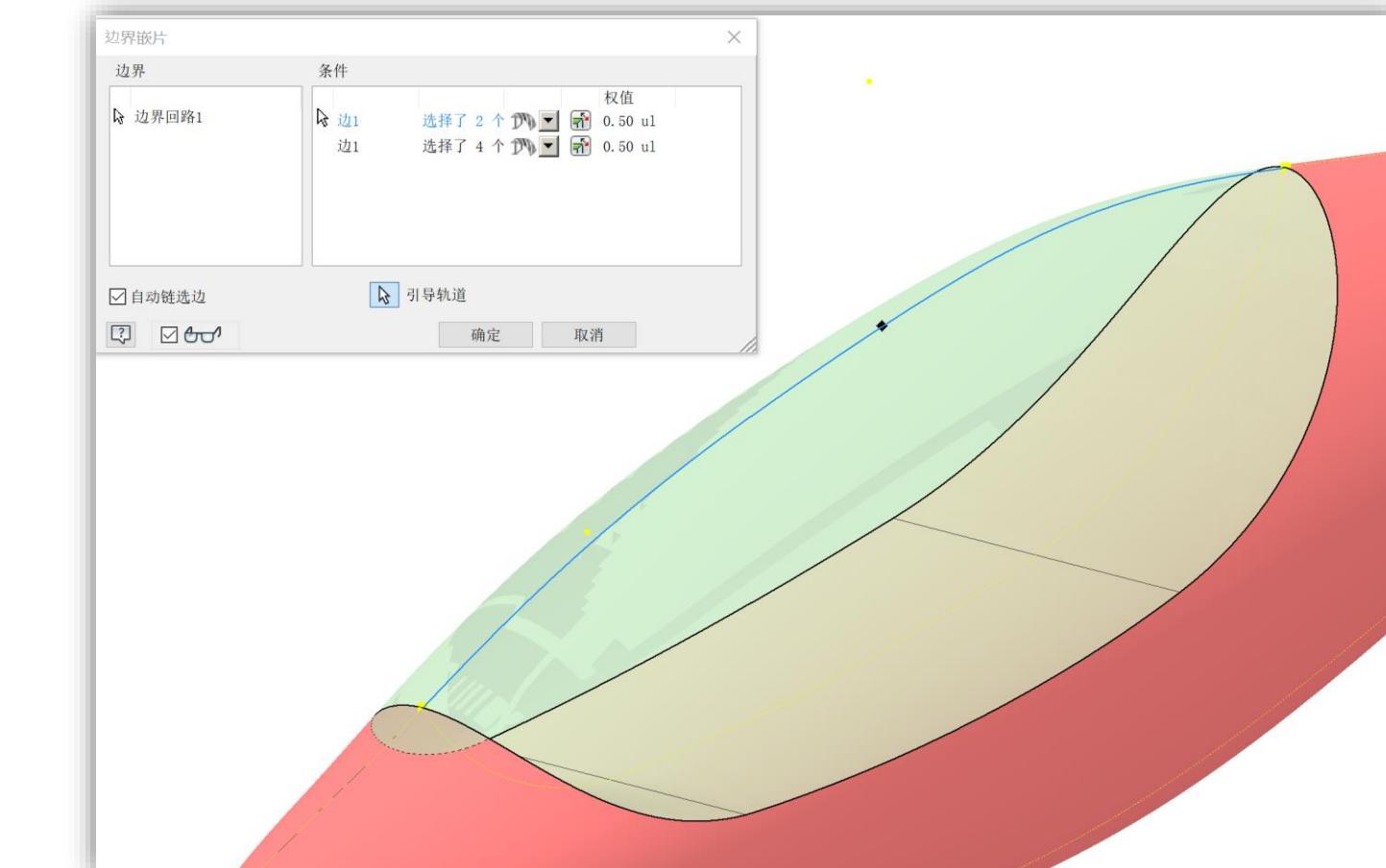
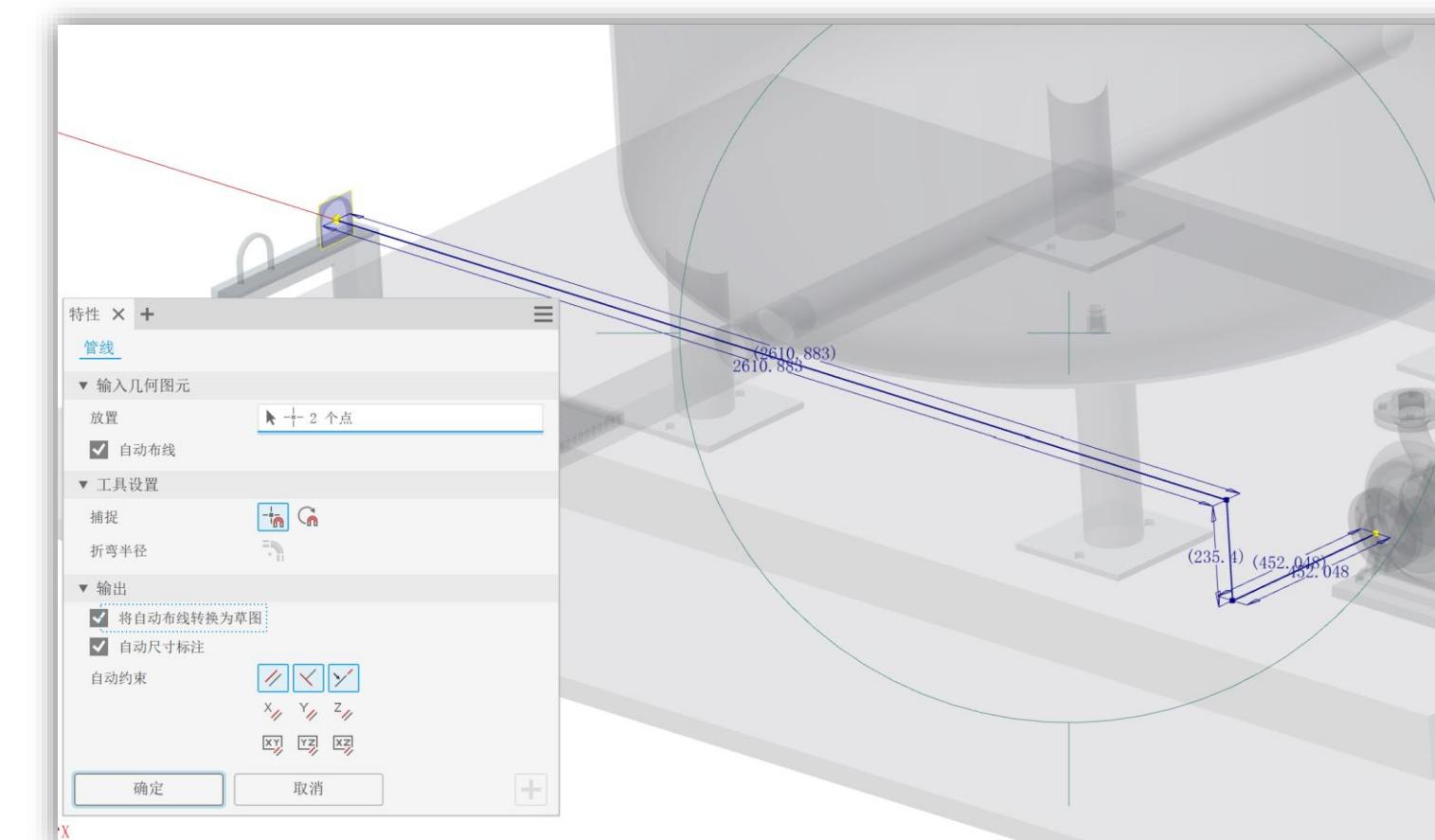
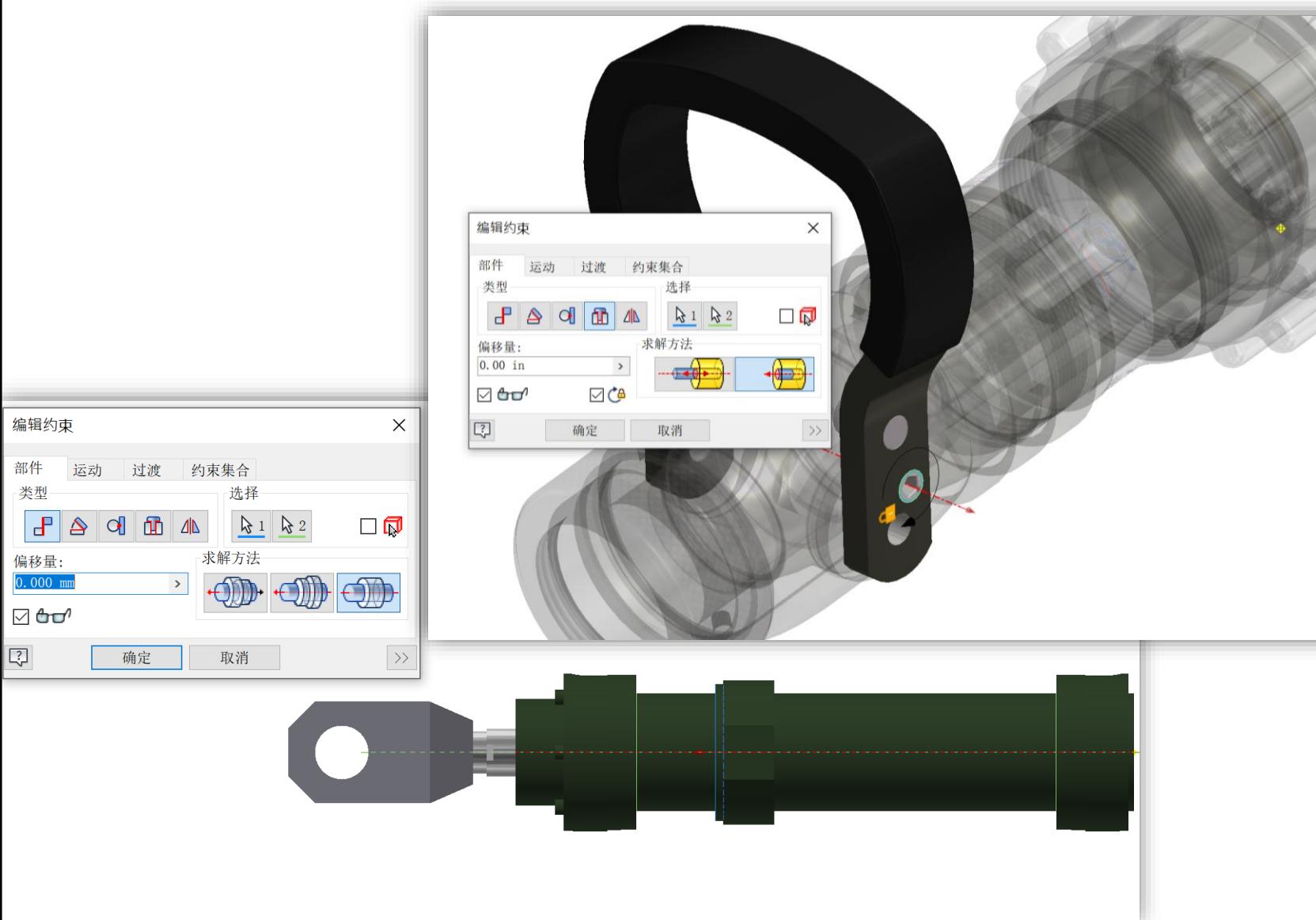
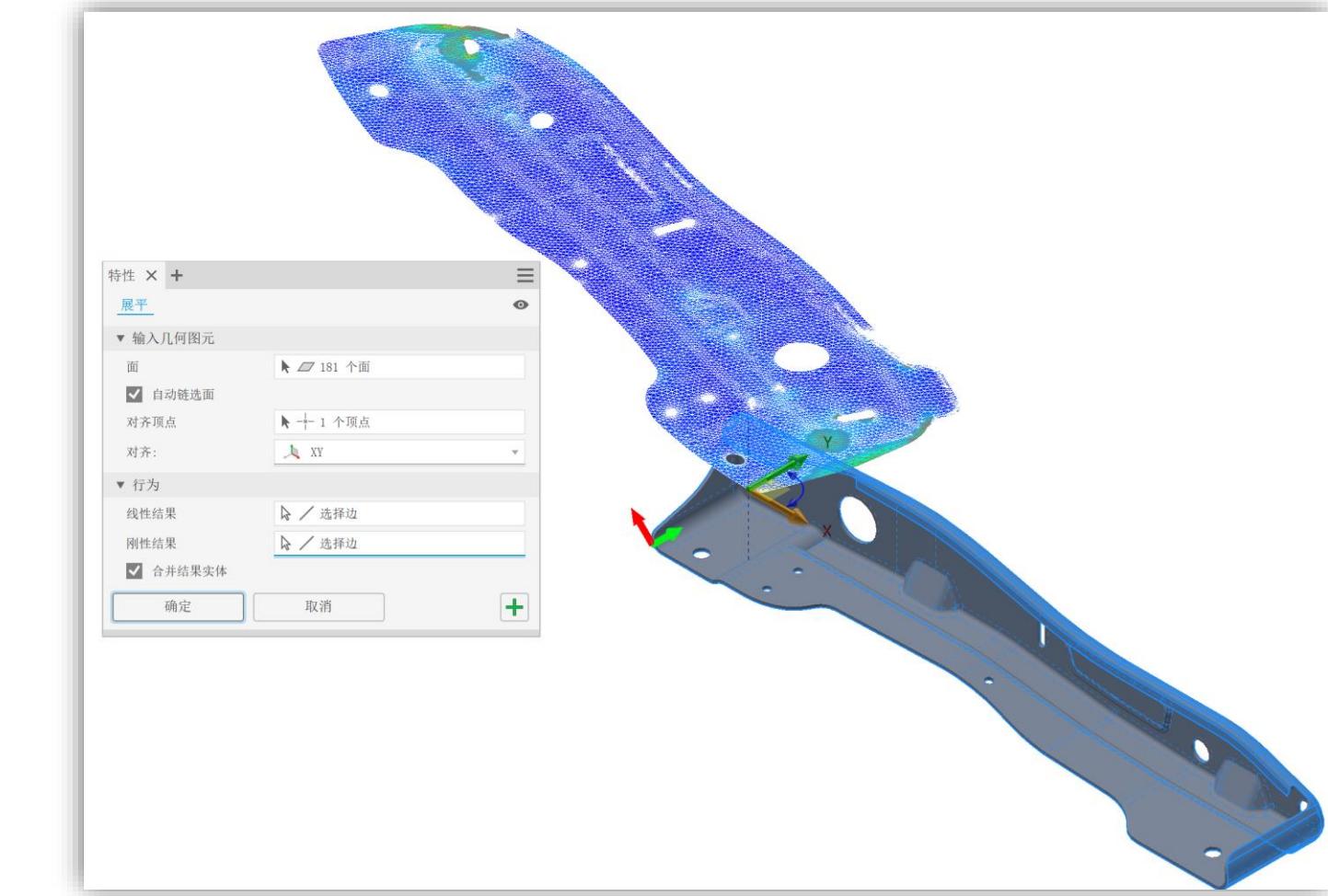
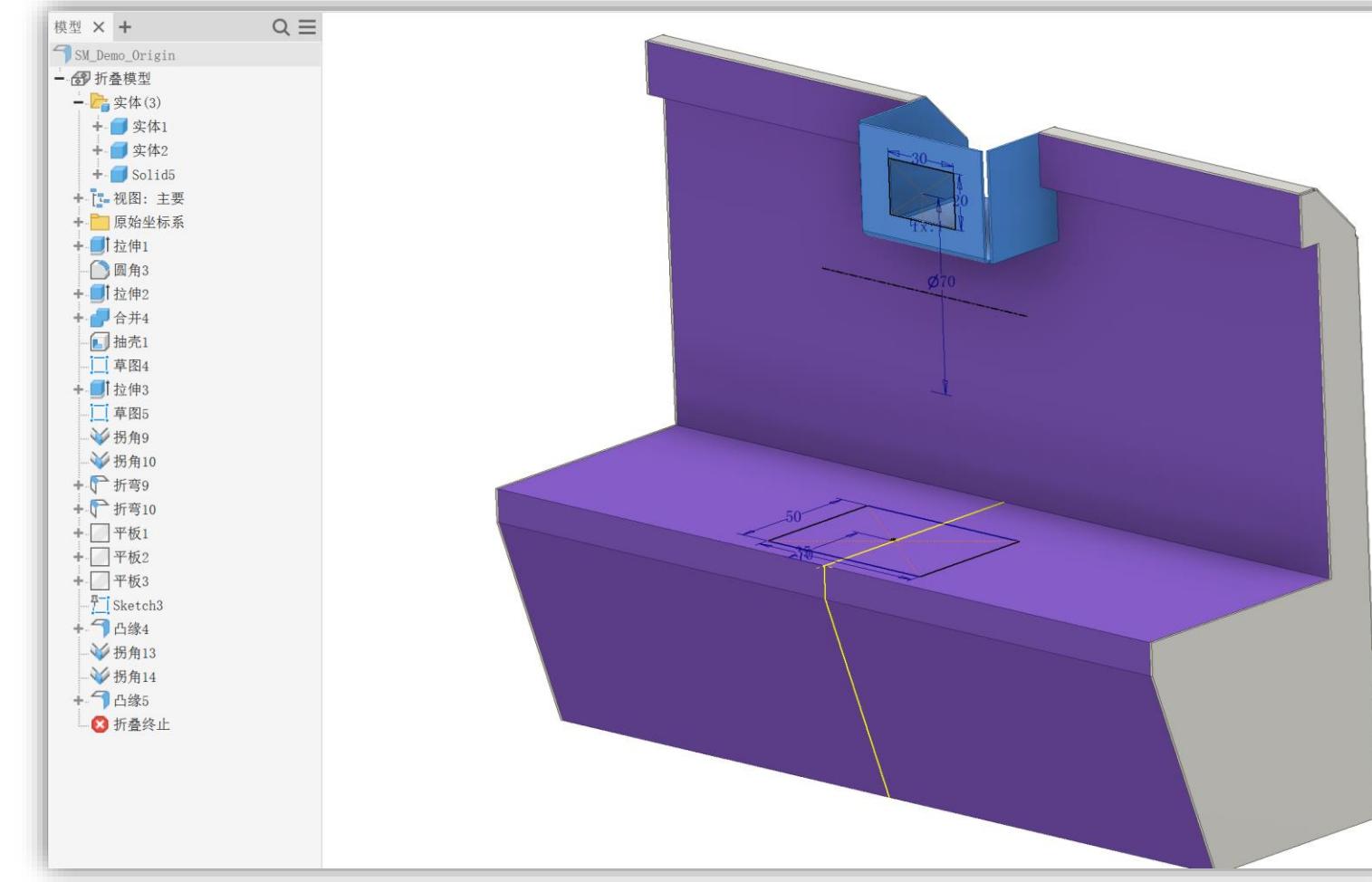
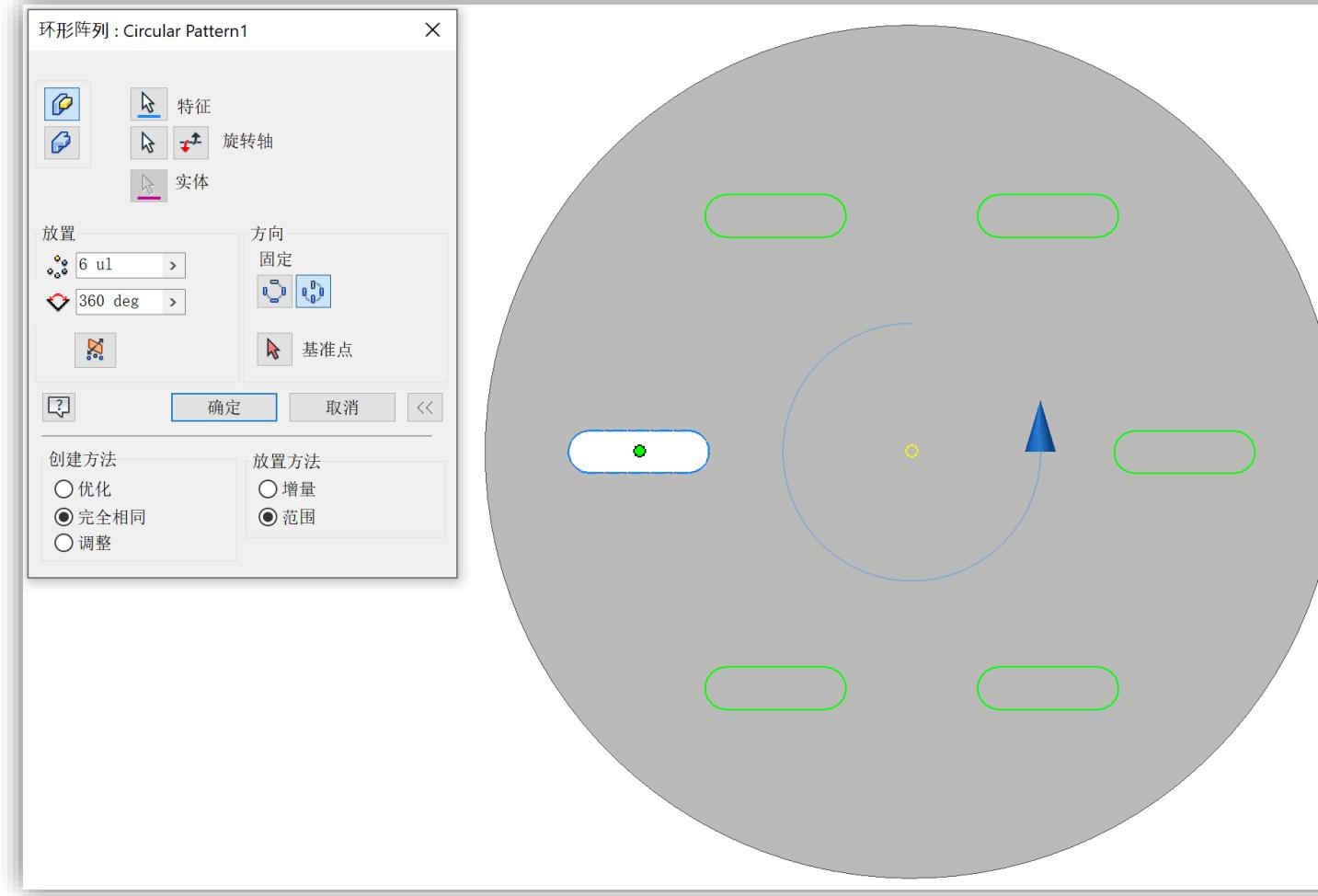
Add tags

VOTE 61 Votes

6 Comments (6 New) POST A COMMENT



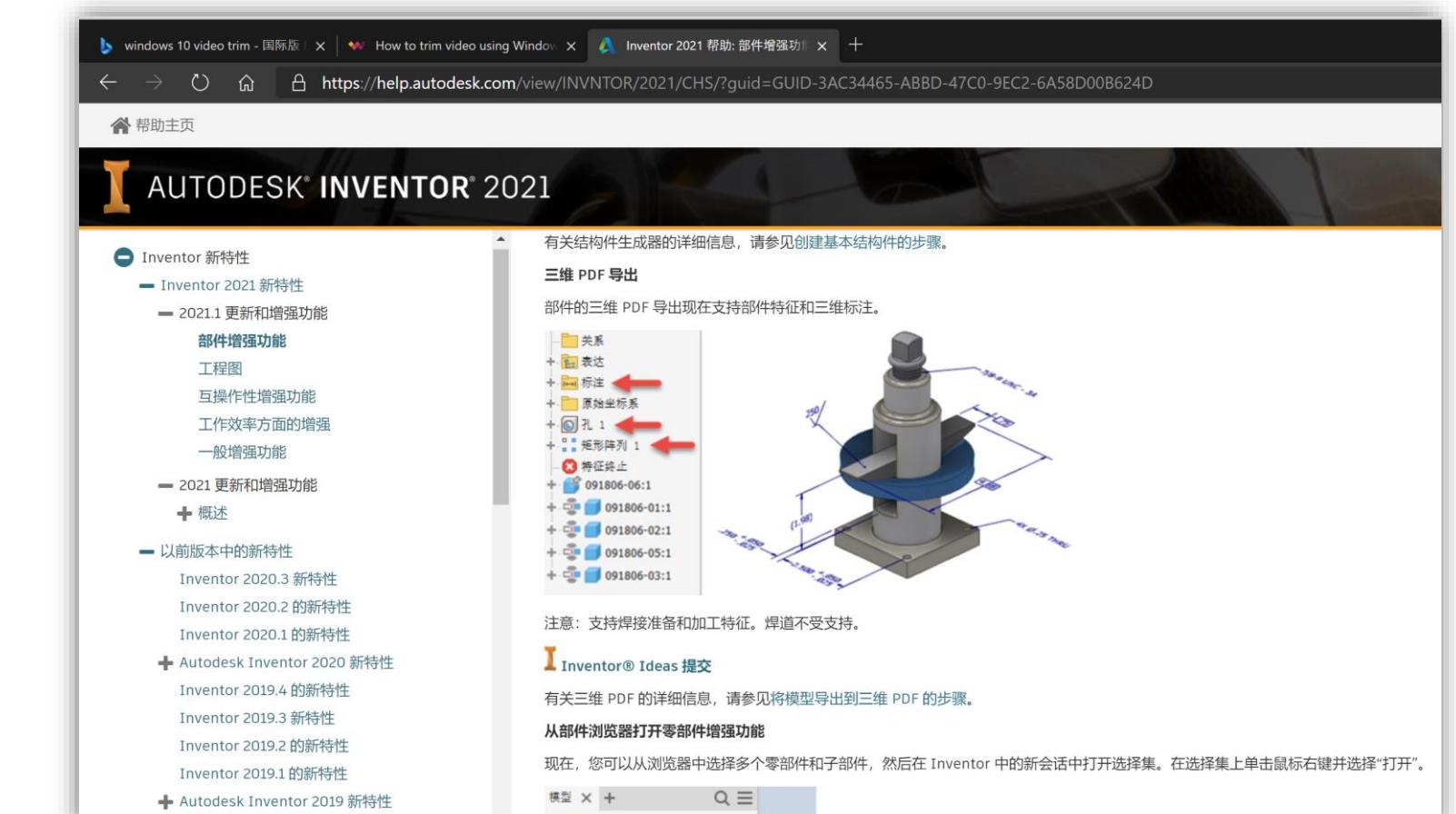
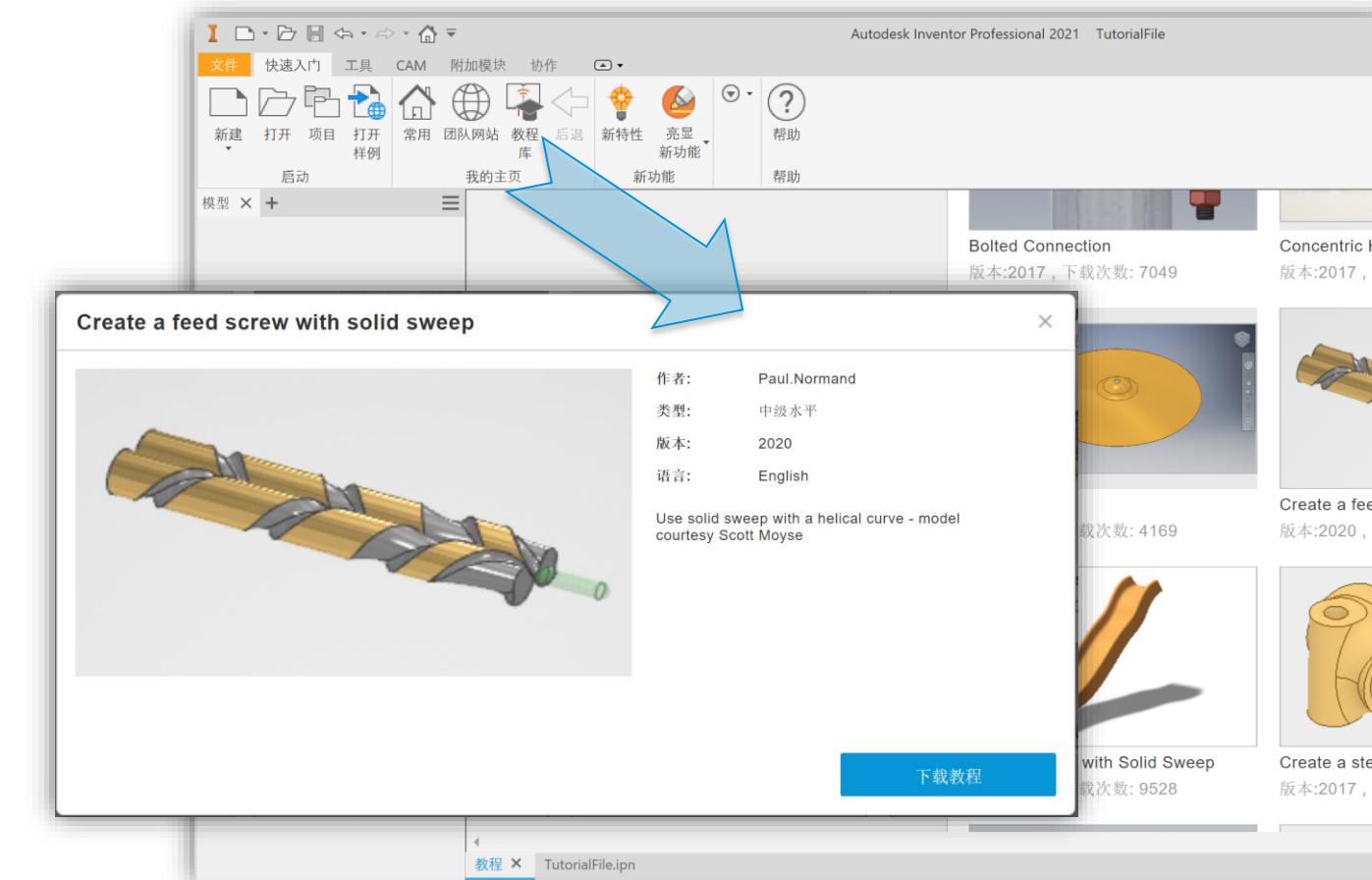
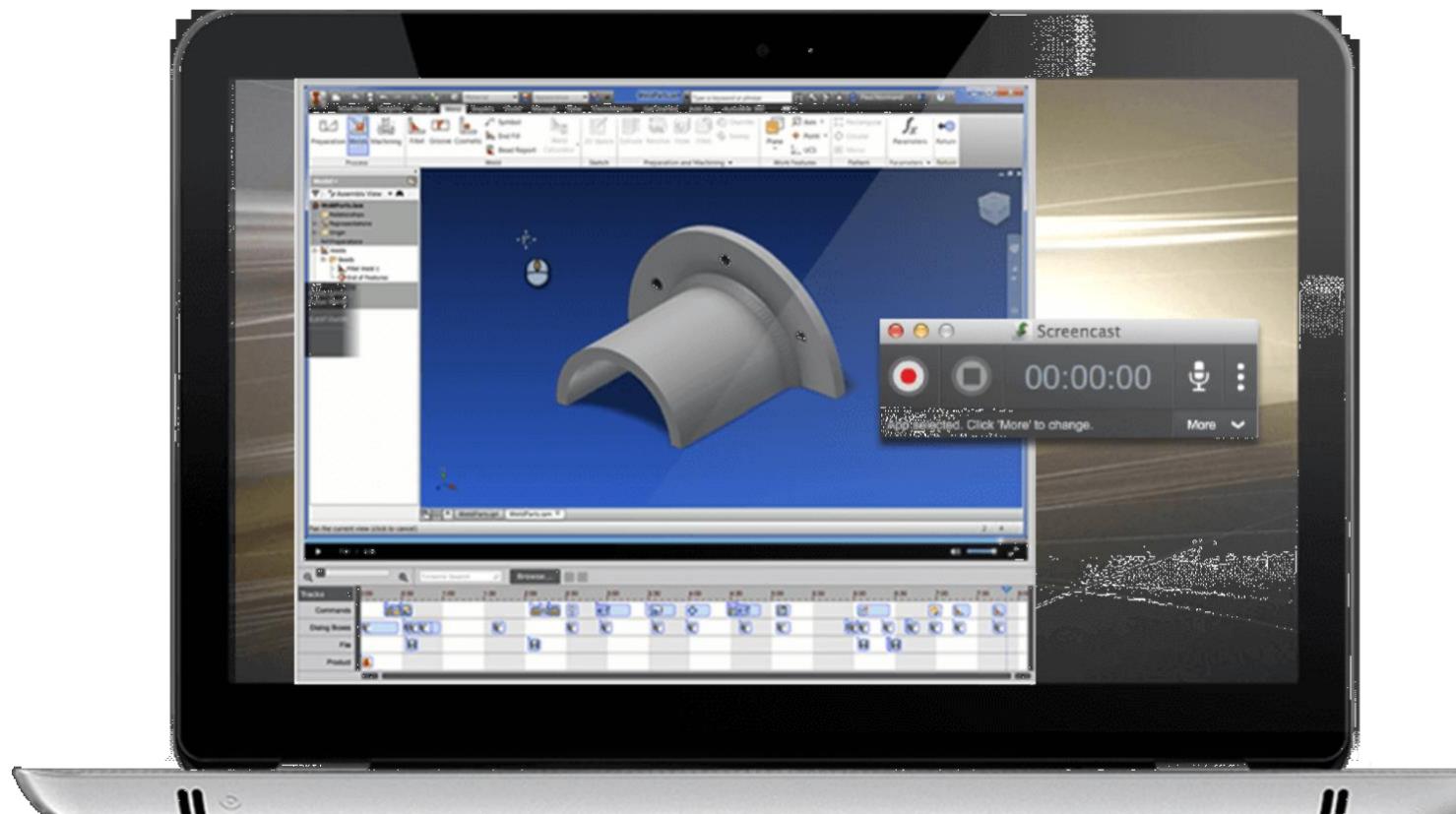
增强内容涉及Inventor的各个方面



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 Forums 论坛
 - Inventor Ideas 用户金点子
- <https://forums.Autodesk.com>
- Inventor 产品技术应用讨论区 ([链接](#))
 - Inventor 反馈社区 ([受邀参加](#))
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-
- 邮件: mfg.usergroup.china@autodesk.com



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- [参考件在工程图的多种显示方式](#)
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