



Training Manual



# 输入几何模型



- 说明
- 将下面 CAD模型输入到 ANSYS:

3. 输入几何模型







Training Manual

## 练习 3A 输入几何模型

## IGES

### 3A.*输入几何模型* IGES





O'LL STAND

1. 按教师指定的工作目录,用"iges"作为作业名,进入ANSYS。 2. 用缺省的设置输入 "bracket.igs" IGES文件: • 选择 "No defeaturing", 然后按 [OK] 选择 "bracket.igs",然后按 [OK] • ▲Import IGES File X 选择 "bracket.igs",然后按 [Open] • [/AUX15] [IOPTN] Options for IGES Import 按[OK] Iges Import Option No defeaturing 或用命令: C Defeature model **/AUX15** <IGESIN> (AUX15 MERGE Merge coincident keypts? Ves Yes X **IOPTN, IGES, NODEFEAT** SOLID Create solid if applicable 🔽 Yes Look jn: 🔁 Intro1 SMALL Delete small areas? ✓ Yes **IGESIN**, bracket, igs 🗋 plate-3 . Date-ОK Cancel Help 🛅 base. **ΙΛ** block 1 🖺 block.err 폐 iges.page 🔊 c-rod-fix.db1 🛋 cutter.igs 🖺 cutter-area.inp 🔊 impeller.db1 🗐 block.log Desktop 🗓 knob.sat **\**(%)%)%) bracket.igs 🗓 file.err 🛅 bracket.x\_t 🖺 left-bracket.x\_t bracket-3d.db1 🗒 file.log 폐 menust.tmp My Computer 🛅 multi-Is.inp 🖌 conn-rod.db1 🖺 qear.sat <u>a -</u> 🔊 conn-rod.db2 p-block.db1 ∧ hextet.db ▲Import IGES File Þ • GTOLER Tolerance for merging Use default • bracket.igs Ŧ File name: Open [IGESIN] (AUX15) File to import Browse. -(\*.\*) Files of type: Cancel OK Help Cancel

### 3A.输入几何模型 IGES



#### 3.保存数据库:

- 在工具栏上按 "SAVE\_DB"按钮 (或选择: Utility Menu > File > Save as Jobname.db)

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Training Manual

练习 **3B** 输入几何模型



### 3B.*输入几何模型* **SAT**



OK.

Cancel

Help

Network..

#### 1.清除ANSYS的数据库:

- Utility Menu > File > Clear & Start New ...
  - [OK],然后按 [Yes]
- 2. 改作业名为 "sat":
  - Utility Menu > File > Change Jobname ...
    - 键入 "sat"作为新的作业名, 然后按 [OK]
- 3.输入 "gear.sat" SAT文件:
  - Utility Menu > File > Import > SAT ...
    - 选择 "gear.sat" 文件, 然后按 [OK]
  - 或用命令:
    - ~SATIN,gear,sat,,SOLIDS,0
- 4.打开 "Normal Faceting":
  - Utility Menu > PlotCtrls > Style > Solid Model Facets ...
    - 选择"Normal Faceting",然后按 [OK]
  - Utility Menu > Plot > Replot
  - 或用命令:
    - /FACET,NORML
    - /REPLOT



ANSYS Connection for SAT

Directories:

/\c

Drives:

🔳 c:

Geometry Type:

Solids Only

•

c:\courses\intro1

🕞 courses

📂 Intro1

plate-13c report

plate-13c\_report

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-

File Name

gear.sat

gear.sat knob.sat

plate.sat

socket.sat

List Files of Type:

Allow Defeaturing

Part File (\*.sat)

crank-assy.sat

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1.1	~
	C
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	-
	3
	0



### 3B.输入几何模型 **SAT**



#### 5.保存数据库:

– 在工具条上按 "SAVE\_DB"按钮 (或选择: Utility Menu > File > Save as Jobname.db)





Training Manual

## 练习 3C 输入几何模型

# SAT集合

### 1.清除ANSYS的数据库:

- Utility Menu > File > Clear & Start New ...
  - [OK], 然后按 [Yes]
- 2.改作业名为 "sat-assy":
  - Utility Menu > File > Change Jobname ...
    - 键入 "sat-assy"作为新的作业名, 然后按 [OK]
- 3.输入 "crank-assy.sat" SAT 集合文件:
  - Utility Menu > File > Import > SAT ...
    - 选择 "crank-assy.sat" 文件, 然后按 [OK]
  - 或用命令:
    - ~SATIN,crank-assy,sat,,SOLIDS,0

#### 4.打开 "Normal Faceting":

- Utility Menu > PlotCtrls > Style > Solid Model Facets ...
  - 选择 "Normal Faceting",然后按 [OK]
- **Utility Menu > Plot > Replot**
- 或用命令:
  - /FACET,NORML
  - **/REPLOT** •





### 3C.输入几何模型 **SAT**集合

### 3C. 输入几何模型 SAT集合





#### 5.打开实体编号开关:

- Utility Menu > PlotCtrls > Numbering ...
  - 将实体编号开关设为 "on", 然后按 [OK]
- Utility Menu > Plot > Volumes
- 或用命令:
  - /PNUM,VOLU,1 VPLOT

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ISYS Toolbar		
WE_DB RESUM_DB QUIT POW	RGRPH	
VSYS Main Menu Preferences Preprocessor Solution General Postoric General Postoric General Postoric Postoric Postoric Design Opt Prob Design Redutation Opt Redutation Opt Session Editor Finish	VOLUMES     VOLU NUM	
	File: crank-assy	



### 3C.输入几何模型 SAT集合



#### 3.保存数据库:

- 在工具条上拾取 "SAVE\_DB" (或选择: Utility Menu > File > Save as Jobname.db)

#### 7.清除ANSYS的数据库:

- Utility Menu > File > Clear & Start New ...
  - [OK], 然后按 [Yes]

#### 8.把工作文件名改为 "sat-assy2":

- Utility Menu > File > Change Jobname ...
  - 键入新工作文件名 "sat-assy2",然后按 [OK]

#### 9.通过输入单个的sat 文件组成的集合来输入曲柄集合:

- Utility Menu > File > Import > SAT ...
  - 选择 "knob.sat" 文件, 然后按 [OK]
- Utility Menu > File > Import > SAT ...
  - 选择 "plate.sat" 文件, 然后按 [OK]
- Utility Menu > File > Import > SAT ...
  - 选择 "socket.sat" 文件, 然后按 [OK]
- 或用命令:
  - ~SATIN,knob,sat,,SOLIDS,0
  - ~SATIN,plate,sat,,SOLIDS,0
  - ~SATIN,socket,sat,,SOLIDS,0



#### 10.打开"Normal Faceting":

3C. 输入几何模型

**SAT**集合

- Utility Menu > PlotCtrls > Style > Solid Model Facets ...
  - 选择 "Normal Faceting",然后按 [OK]
- Utility Menu > Plot > Replot
- 或用命令:

/FACET,NORML /REPLOT

#### 11.打开实体编号开关:

- Utility Menu > PlotCtrls > Numbering ...
  - 将实体编号开关设为 "on",然后按 [OK
- Utility Menu > Plot > Volumes
- 或用命令:
  - /PNUM,VOLU,1 VPLOT



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Training Manual

### 练习 3D 输入几何模型

# Parasolid

### 3D.输入几何模型 Parasolid





#### 1.清除 ANSYS 数据库:

- Utility Menu > File > Clear & Start New ...
  - [OK], 然后按 [Yes]

#### 2.改作业名为 "para":

- Utility Menu > File > Change Jobname ...
  - 键入 "para" 作为新的作业名, 然后按 [OK]

#### 3.输入 "bracket.x\_t" 实体参数文件:

- Utility Menu > File > Import > PARA ...
  - 选择 "bracket.x\_t" 文件, 然后按 [OK]
- 或用命令:
  - ~PARAIN,bracket,x\_t,,SOLIDS,0,0

#### 4.打开"Normal Faceting":

- Utility Menu > PlotCtrls > Style > Solid Model Facets ...
  - 选择 "Normal Faceting", 然后按 [OK]
- Utility Menu > Plot > Volumes
- 或用命令:

/FACET,NORM

VPLOT





### 3D.输入几何模型 Parasolid



#### 5.保存数据库:

- 在工具条上按 "SAVE\_DB"按钮 (或选择: Utility Menu > File > Save as Jobname.db)

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	र रहेकारका होत्र के लिए	
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AVE_DB RESUM_DB QUIT POWRGRP	4	
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Training Manual

## 练习 3E 输入几何模型

# Parasolid 集合

### 3E.输入几何模型 Parasolid集合





- Utility Menu > File > Clear & Start New ...
  - [OK],然后按 [Yes]
- 2.改变工作名为"para-assy":
  - Utility Menu > File > Change Jobname ...
    - 键入 "para-assy"作为新的工作名, 然后按 [OK]
- 3.输入"wheel-assy.x\_t" 实体参数集合文件:
  - Utility Menu > File > Import > PARA ...
    - 选择 "wheel-assy.x\_t" 文件, 然后按 [OK]
  - 或用命令:
    - ~PARAIN,wheel-assy,x\_t,,SOLIDS,0,0

#### 4.打开 "Normal Faceting":

- Utility Menu > PlotCtrls > Style > Solid Model Facets ...
  - 选择 "Normal Faceting",然后按 [OK]
- Utility Menu > Plot > Replot
- 或用命令:
  - /FACET,NORML /REPLOT



### 3E.输入几何模型 Parasolid集合





#### 5.打开实体编号开关:

- Utility Menu > PlotCtrls > Numbering ...
  - 将实体编号开关设为 "on",然后按 [OK]
- Utility Menu > Plot > Volumes
- 或用命令:
  - /PNUM,VOLU,1 VPLOT

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3.保存数据库:

- 在工具条上按 "SAVE\_DB" (或选择: Utility Menu > File > Save as Jobname.db)

7.清除ANSYS 数据库:

- Utility Menu > File > Clear & Start New ...
  - [OK],然后按 [Yes]

8.改工作名为 "para-assy2":

- Utility Menu > File > Change Jobname ...
  - 键入新工作名 "para-assy2",然后按 [OK]







9.通过输入组成集合的单个实体参数文件来输入齿轮集合:

- Utility Menu > File > Import > PARA ...
  - 选择 "base.x\_t"文件, 然后 [OK]
- Utility Menu > File > Import > PARA ...
  - 选择 "left-bracket.x\_t"文件, 然后按 [OK]
- Utility Menu > File > Import > PARA ...
  - 选择 "right-bracket.x\_t" 文件, 然后按 [OK]
- Utility Menu > File > Import > PARA ...
  - 选择 "pin.x\_t" 文件, 然后按 [OK]
- Utility Menu > File > Import > PARA ...
  - 选择 "wheel.x\_t"文件, 然后按 [OK]
- 或用命令:
  - ~PARAIN,base,x\_t,,SOLIDS,0,0
  - ~PARAIN,left-bracket,x\_t,,SOLIDS,0,0
  - ~PARAIN,right-bracket,x\_t,,SOLIDS,0,0
  - ~PARAIN,pin,x\_t,,SOLIDS,0,0
  - ~PARAIN,wheel,x\_t,,SOLIDS,0,0



#### 10.打开"Normal Faceting":

3E.输入几何模型

Parasolid集合

- Utility Menu > PlotCtrls > Style > Solid Model Facets ...
  - 选择"Normal Faceting", 然后按 [OK]
- Utility Menu > Plot > Replot
- \_ 或用命令:

/FACET,NORML

/REPLOT

#### 11.将实体编号开关打开:

- Utility Menu > PlotCtrls > Numbering ...
  - 将实体编号开关设为 "on",然后按 [OK]
- Utility Menu > Plot > Volumes
- 或用命令:

/PNUM,VOLU,1

VPLOT

#### 12.退出 ANSYS:

- 按 "QUIT"
- 选择 "Quit No Save!"
  - [OK]
- 或用命令:

FINISH \$ /EXIT,NOSAVE



File: C:\Courses\Introl\wheel.x









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Training Manual

### 专题 3F 实体建模: 自上而下

# 轴承座



Bushing, 0.85R



实体模型。

### 3F. 实体建模: 自上而下 轴承座

说明

- 建立轴承座的1/2对称



Bracket

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### 1.按教师指定的工作目录,用"p-block"作为作业名,进入ANSYS

2.打开等轴视图方位:

承应

• 按[ISO]

3F. 实体建模: 自上而下

- 或用命令: /VIEW,1,1,1,1
- 3.创建轴承座的基础:
  - Main Menu > Preprocessor > Modeling > Create > Volumes > Block > By Dimensions
    - 输入 X1 = 0, X2 = 3, Y1 = 0, Y2 = 1, Z1 = 0, Z2 = 3, 然后按[OK]
  - 或用命令:
    - /PREP7
    - BLOCK,0,3,0,1,0,3

Create Block by Dimensions	×
[BLOCK] Create Block by Dimensions	
X1,X2 X-coordinates	0 3
Y1,Y2 Y-coordinates	0 1
Z1,Z2 Z-coordinates	0 3
OK Apply	Cancel Help





### 3F. 实体建模: 自上而下 **轴承座**

#### 4.将工作平面移到 X=2.25, Y=1.25, Z=.75:

- Utility Menu > WorkPlane > Offset WP by Increments ...
  - 设置 X,Y,Z Offsets = 2.25, 1.25, 0.75
  - 设置XY, YZ, ZX Angles = 0, -90, 0, 然后按 [OK]
- 或用命令:

WPOFF, 2.25, 1.25, 0.75 WPROT, 0, -90, 0



Offset W	/Ρ
X-	+X
<u>Z-</u>	+ <u>Z</u>
Snaps	<u> </u>
X.YZ Offset	s
2.25,1.25,0	.75
X- 0	र्भ <del>भ</del>
Y- 0	Ð+Y
Z-Q	Ð+Z
30	
	Þ
Degrees	
XY, YZ, ZX	Angles
0,-90,0	
Global X=	0
Y=	0
Z=	0
🗖 Dynami	c Mode
OK	Apply
Beset	Cancel
Неір	



Training Manual



#### Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder 输入Radius = 0.75/2 • • 输入Depth = -1.5, 然后按 [OK] Solid Cylinder X 或用命令: \_ Pick C Unpick • CYL4, ., 0.75/2, ., .-1.5 WP X ¥. 3.将圆柱考贝到DZ=1.5的新位置: Global X = Main Menu > Preprocessor > Modeling > Copy > Volumes + Y = 拾取柱体 (体号 2),按 [OK] • Z = • DZ = 1.5, 按 [OK] WP X 或用命令: \_ Copy Volumes $\times$ WP Y [VGEN] Copy Volumes • VGEN,2,2,...,1.5,.0 Radius 0.75/2 ITIME Number of copies -- including original Depth -1.5 DX X-offset in active CS DY Y-offset in active CS **OK** Apply DZ Z-offset in active CS 1.5 Reset Cancel KINC Keypoint increment Help NOELEM Items to be copied Volumes and mesh ٠ OK Help Apply Cancel



5.创建直径为0.75 英寸深度为-1.5 英寸的圆柱:



היעודיטוריוויטוע נעיד איזערייט איזיאיז



3F. 实体建模: 自上而下

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Volumes
  - 拾取轴承座基础的体(体1),按[OK] •
  - 拾取两个圆柱体 (体 2 和体3), 然后按 [OK] ٠
- 或用命令:

轴承座







### 3F. 实体建模: 自上而下 **轴承座**

#### 8.在整体坐标系中改变工作平面的相对位置:

- Utility Menu > WorkPlane > Align WP with > Global Cartesian
- 或用命令:
  - WPCSYS,-1,0

**VPLOT** 

#### 9.创建套筒托架的基础:

- Main Menu > Preprocessor > Modeling > Create > Volumes > Block > By 2 Corners & Z
  - WP X = 0
  - WP Y = 1
  - Width = 1.5
  - Height = 1.75
  - Depth = 0.75, then [OK]
- 或用命令:BLC4,0,1,1.5,1.75,0.75

#### 10.将工作平面移到套筒托架的正面:

- Utility Menu > WorkPlane > Offset WP to > Keypoints +
  - 拾取正面左角顶部的关键点,按 [OK]
- 或用命令:

KWPAVE, 13



Block by 2 Corner.. Pick
 C Unpick WP X Global X = Y = Z =WP X WP Y Width 1.5 Height 1.75 Depth 0.75 OK Apply Cance1 Reset Help





#### 11.创建套筒托架的拱:

轴承座

\_

- Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Partial Cylinder
  - 输入WP X = 0

3F. 实体建模: 自上而下

- 输入WP Y = 0
- 输入Rad-1=0
- 输入Theta-1 = 0
- 输入Rad-2 = 1.5
- 输入Theta-2 = 90
- 输入Depth = -0.75, 然后按 [OK]
- 或用命令:
  - CYL4,0,0,0,0,1.5,90,-0.75





#### 12.通过套筒托架的孔创建圆柱:

- Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder
  - WP  $\mathbf{X} = \mathbf{0}$
  - WP Y = 0
  - Radius = 1
  - Depth = -0.1875,然后按 [Apply]
  - WP X = 0
  - WP Y = 0
  - **Radius** = 0.85
  - Depth = -2, 然后按 [OK]
- 或执行:
  - CYL4,0,0,1, , , ,-0.1875 CYL4,0,0,0.85, , , ,-2











#### 13.挖掉两个圆柱,形成轴承座和套筒的孔:

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Volumes
  - 拾取两个形成套筒托架拱和基础的体
  - 按[Apply]
  - 拾取轴承座柱
  - 按[Apply]
  - 拾取同样的两个基础的体
  - 按[Apply]
  - 拾取通过孔的圆柱
  - 按[OK



14.合并相同的关键点:

- Main Menu > Preprocessor > Numbering Ctrls > Merge Items
  - 设置标号 "Keypoints",然后按 [OK]
- 或用命令:

NUMMRG,KP

ANSYS 8.0 Output Window	<u>_ 🗆 ×</u>
MERGE COINCIDENT KEVPOINTS WITHIN TOLERANCE OF KEVPOINT         0.10000           KEVPOINT         1 USED FOR KEVPOINT(S)         10           KEVPOINT         11 USED FOR KEVPOINT(S)         18           KEVPOINT         15 USED FOR KEVPOINT(S)         18           KEVPOINT         15 USED FOR KEVPOINT(S)         50           LINE         47 USED FOR KEVPOINT(S)         50           LINE         47 USED FOR LINE(S)         32           LINE         23 USED FOR KEVPOINT(S)         50           LINE         23 USED FOR KEVPOINT(S)         50           LINE         23 USED FOR KEVPOINT(S)         52           LINE         19 USED FOR KENPOINT(S)         52           LINE         23 USED FOR KENPOINT(S)         52           LINE         19 USED FOR KENPOINT(S)         32           LINE         19 USED FOR KENE(S)         32           LINE         19 USED FOR KENE(S)         52           LINE         68 USED FOR KENE(S)         76           AREA         14 USED FOR AREA(S)         25	9E-03
4	▶ //.





### 3F. 实体建模: 自上而下 轴承应







## 3F. 实体建模: 自上而下 **轴承座**

Training Manual

- D X

#### 15b.创建三角形面:

- Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs
  - 在轴承座基础和套筒托架基础相交的位置, 拾取第一个关键点 X=1.5
  - 在拱表面底部和套筒托架基础相交的位置, 拾取第二个关键点 X=1.5
  - 拾取在14a步骤中建立的位于X=1.5, Y=1, Z=3的第三个关键点

ANSYS Multiphysics Utility Menu (p-block)

- 按[Ok]
- 或用命令:

A,14,15,9



#### Main Menu > Preprocessor > Modeling > Operate > Extrude > Areas Along Normal \_ 拾取在步骤14b中建立的三角形面,按[OK] • 输入DIST = -0.15, 然后按[OK] • 或用命令: \_ Extrude Area along Normal **VOFFST,3,-0.15** [VOFFST] Extrude Area along Normal NAREA Area to be extruded ANSYS Multiphysics Utility Menu (p-block) - 0 × Elle Select List Plot PlotOtis WorkPlane Pagameters Macro MeguCtris Help DIST Length of extrusion -0.15 D 📽 🖬 🖲 🗃 🖉 🖻 📓 • 🗉 💷 ANSYS Toolbar KINC Keypoint increment SAVE\_DB RESUM\_DB OUT POWRGRPH ANSYS Main Me Preferences Preprocessor Element Type Real Constants ANS VOLUMES OK. Apply Cancel Help TYPE NUM Material Props Sections E Create E Operate E Extrude Elem Ext Opts Along Norma About Avis W2 WX Along Line B Lines B Keypoints B Extend Line B Booleans Biscalar Biscalar Biscalar Mover / Modify Biscalar Biscal E Scale E FLOTRAN Set Up E FSI Set Up MultiField Set Up I Loads Physics E Path Operations Solution Solution General Postproc TimeHist Postpro Topological Opt ROM Tool HOM Tool Design Opt Prob Design Prob Design Radiation Opt Run-Time Stats Session Editor Finish Pick a menu item or enter an ANSYS Command (PREP7) mat-1 type-1 real-1

3F. 实体建模: 自上而下 **轴承座** 

15c.沿面的法线方向拉伸面:

Training Manual





### 13.组合体:

THE ISTA

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes
  - 拾取[Pick All] •
- 或用命令:

**VGLUE,ALL** 

3F. 实体建模: 自上而下

#### 17.打开体号的显示开关, 画体:

- Utility Menu > PlotCtrls > Numbering ...
  - 设置 Volume numbers选项为 on, 按 [OK]
- 或用命令:

/PNUM,VOLU,1 **VPLOT** 

#### 18.保存数据库,退出ANSYS:

- Pick the "SAVE DB" button in the Toolbar
- Pick the "QUIT" button in the Toolbar
  - 选择 "Quit No Save!" ٠
  - 按[OK]
- 或用命令:
  - **FINISH**
  - /EXIT,ALL











Training Manual

### 练习 3G 实体建模:由下而上







#### 1.按教师指定的工作目录,用"c-rod"作为作业名,进入ANSYS。 2.创建两个圆形面:

- Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions
  - 输入RAD1 = 1.4
  - 输入RAD2=1
  - 输入THETA1 = 0
  - 输入THETA2 = 180, 然后选择[Apply]
  - 输入THETA1 = 45, 然后选择[OK]
- 或用命令:
  - /PREP7

3G.实体建模-由下而上

- PCIRC,1.4,1,0,180
- PCIRC,1.4,1,45,180
- 3.打开 面号:
  - Utility Menu > PlotCtrls > Numbering ...
    - 设置面号为 "on", 然后选择[OK]
  - 或用命令:
    - /PNUM,AREA,1
    - APLOT

Circular Area by Dimensions	×
[PCIRC] Circular Area by Dimensions	
RAD1 Outer radius	1.4
RAD2 Optional inner radius	1
THETA1 Starting angle (degrees)	0
THETA2 Ending angle (degrees)	180
OK Apply Cancel	Holp
Circular Area by Dimensions	×
[PCIBC] Circular Area by Dimensions	
RAD1 Outer radius	1.4
BAD2 Optional inner radius	1











#### 4.创建两个矩形面:

- Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions
  - 输入X1 = -0.3, X2 = 0.3, Y1 = 1.2, Y2 = 1.8, 然后选择[Apply]
  - 输入X1 = -1.8, X2 = -1.2, Y1 = 0, Y2 = 0.3, 然后选择[OK]
- 或用命令:

RECTNG,-0.3,0.3,1.2,1.8

RECTNG,-1.8,-1.2,0,0.3

#### 5.平移工作面位置(X=3.5):

- Utility Menu > WorkPlane > Offset WP to > XYZ Locations +
  - 回车后在输入窗口输入 3.5, [OK]
- 或用命令:
- Or issue:

WPAVE,3.5



#### 3.再创建两个圆形面:

连杆

3G.实体建模-由下而上

- Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions
  - 输入RAD1 = 0.7
  - 输入RAD2 = 0.4
  - 输入THETA1 = 0
  - 输入THETA2 = 180, 然后选择[Apply]
  - 输入THETA2 = 135, 然后选择[OK]
- 或用命令:

PCIRC,0.7,0.4,0,180 PCIRC,0.7,0.4,0,135











#### 7.在每一组面上分别进行布尔操作:

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas
  - 先选择左边的一组面, 然后选择[Apply]
  - 再选择右边的一组面,然后选择[OK]
- 或用命令:
  - AOVLAP,1,2,3,4
  - AOVLAP,5,3

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AND TOKE OUT FORMATING	
SAVE_DB_RESUALDB_OUT_POWRAPPH ARCTIS Main Manu @ Preferences 1	
AND'IS Main Mane (B)	
Preferences 2	1
Proprocessor     Provide and Program     Provide and Program     Provide and Program     Provide and Provide	







#### 8.定义四个新的关键点:

- Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS
  - 第一关键点, X=2.5, Y=0.5, 然后选择[Apply]
  - 第二关键点, X=3.25, Y=0.4, 然后选择[Apply]
  - 第三关键点, X=4, Y=0.33, 然后选择[Apply]
  - 第四关键点, X=4.75, Y=0.28, 然后选择[OK]
- 或用命令:
  - K, ,2.5,0.5
  - К, ,3.25,0.4
  - K, ,4.0,0.33
  - K, ,4.75,0.28

9.激活整体柱坐标系:

- Utility Menu > WorkPlane > Change Active CS to > Global Cylindrical
- 或用命令:
  - CSYS,1





- 10. 创建一条线(由一系列关键点拟合一条样条曲线):
  - Main Menu > Preprocessor > Modeling > Create > Lines > Splines > With Options > Spline thru KPs
    - 顺序拾取如图形窗口所示的六个关键点,然后选择[OK]







#### 10. (续):

- 输入XV1=1(总体柱坐标系,关键点1处的半径)
- YV1 = 135 (总体柱坐标系,关键点1处的角度)
- XV3 = 1 (总体柱坐标系,关键点 3 处的半径)
- YV3 = 45 (总体柱坐标系,关键点 3 处的角度)
- 按[OK]
- 或用命令:

#### BSPLIN,5,3,7,21,24,22,1,135,,1,45

▲ B-Spline		X
[BSPLIN] Generate a Spline		
Start tangent XV1, YV1, ZV1	1 135	
Ending tangent XV6,YV6,ZV6	1 45	
OK Apply	Cancel	Help









#### 11.通过关键点1和18创建一条直线:

- Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line
  - 拾取图形窗口所示的两个关键点,然后选择[OK]
- 或用命令:

LSTR, 1, 18







12.打开线号,显示线:

- Utility Menu > PlotCtrls > Numbering ...
  - 设置 Line numbers为 "on", 然后选择[OK]
- Utility Menu > Plot > Lines
- \_ 或用命令:

/PNUM,LINE,1

LPLOT

13.以3号、1号、7号、25号线为边界创建一个新面:

- Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines
  - 拾取四条线 (3, 1, 7, 和 25), 然后选择[OK]
- 或用命令:

AL, 3, 1, 7, 25





14.放大连杆左边部分:

连杆

3G.实体建模-由下而上

- Use the Zoom Model Icon in the Model Control Toolbar

ANSYS Multiphysics Utility Menu (c-rod) \_ 🗆 🗙 Eile Select List Plot PlotQtrls WorkPlane Parameters Macro MeguCtrls Help 🗉 🔳 🖬 D 🛎 🖬 🗗 🗃 🖉 🍸 📓 ANSY'S Toolbar ٢ SAVE\_DB RESUM\_DB OUIT POWRGRPH ANSYS Main Menu 8 1 🗉 Preferences AN • E Preprocessor LINES Element Type
 Real Constants **----**LINE NUM Material Props E Material Props E Sections Modeling E Create E Operate E Move / Modify Copy
Reflect
Check Geom Delete Cyclic Sector
 Genl plane strn
 Update Geom Meshing
 Checking Ctrls
 Numbering Ctrls E Archive Model E Coupling / Ceqn E FLOTNAN Set Up E FSI Set Up MultiField Set Up L25 E Loads 1.36 Physics E Path Operations E Solution L16 33 114 General Postproc
 TimeHist Postpro 132 13 Topological Opt
 ROM Tool Design Opt
 Prob Design Radiation Opt
 Run-Time Stats Session Editor E Finish • Pick a menu item or enter an ANSY'S Command (PREP7) mat-1 type=1 real=1 CSYS=1



PO G





#### 15.创建线与线的倒角:

连杆

\_

- Main Menu > Preprocessor > Modeling > Create > Lines > Line Fillet
  - 拾取线 33 和 40, 然后选择[Apply]
  - 输入RAD = .25, 然后选择[Apply]
  - 拾取线 40和 31, 然后选择[Apply]
  - 按[Apply]

3G.实体建模-由下而上

- 拾取线 30和 39, 然后选择[OK]
- 按[OK]
- Utility Menu > Plot > Lines
- 或用命令:

LFILLT,33,40,0.25 LFILLT,40,31,0.25 LFILLT,30,39,0.25 LPLOT

∧Line Fillet	×
[LFILLT] Create Fillet Line	
NL1,NL2 Intersecting lines	36 40
RAD Fillet radius	0.25
PCENT Number to assign -	
- to generated keypoint at fillet center	
OK Apply	Cancel Help





#### 13.以倒角线为边界,创建新面:

- Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines
  - 拾取线 12, 10, 和 13, 然后选择[Apply]
  - 拾取线 17, 15, 和19, 然后选择[Apply]
  - 拾取线 23, 21, 和24, 然后选择[OK]
- Utility Menu > Plot > Areas
- 或用命令:
  - AL, 12, 10, 13
  - AL, 17, 15, 19
  - AL, 23, 21, 24
  - APLOT







- Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas
  - 拾取[Pick All]
- 或用命令:

AADD,ALL

#### 18.选择Fit 使整个模型充满图形窗口:

- Utility Menu > PlotCtrls > Pan, Zoom, Rotate ...
  - 按[Fit]

#### 19.关闭线号和面号:

- Utility Menu > PlotCtrls > Numbering ...
  - 设置线号和面号为" off ",然后选择[OK]
- Utility Menu > Plot > Areas
- 或用命令:
  - /PNUM,LINE,0 /PNUM,AREA,0
  - APLOT





#### 20.激活整体直角坐标系:

连杆

- Utility Menu > WorkPlane > Change Active CS to > Global Cartesian
- 或用命令:
  - CSYS,0

3G.实体建模-由下而上

#### 21.以X-Z 平面 (在 Y 方向) 为对称面,对面作镜面反射:

- Main Menu > Preprocessor > Modeling > Reflect > Areas
  - [Pick All]
  - Select X-Z plane, then [OK]
- 或用命令:

ARSYM,Y,13

-
×









#### Training Manual

#### 22.把所有的面加起来:

- Main Menu > Preprocessor > Modeling > Operate > Booleans >Add > Areas
  - [Pick All]
- 或用命令:

AADD,ALL

#### 23.关闭工作平面:

- Utility Menu > WorkPlane > Display Working Plane
- 或用命令:

WPSTYLE





#### 24.存储数据库,退出 ANSYS:

- 在工具条中拾取 "SAVE\_DB"
- 在工具条中拾取 "QUIT"
  - 选择 "Quit No Save!"
  - 按[OK]
- 或用命令:

SAVE

FINISH

/EXIT,NOSAVE







练习3H 实体建模:输入/修正

连杆

### 3H.实体建模: 输入/修正







说明

- 对一个连杆的CAD 模型进行分析。
- 这个几何模型并不 包括分析所需的所 有特征。
- 需要添加倒角、孔 并创建1/2对称模型
   来简化分析。









## 3H.实体建模: 输入/修正 连杆



#### 3c.创建一个圆柱体,为生成孔作准备:

- Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder
  - 输入Radius = 1, Depth = -1, 按 [OK]
- 或用命令:

CYL4, , ,1, , , ,-1

#### 3d.从连杆中减去圆柱体:

- Main Menu > Preprocessor > Operate > Booleans > Subtract > Volumes +
  - 拾取 连杆 , 按[OK]
  - 拾取圆柱体, 按[OK] Utility Menu > Plot > Volumes
- 或用命令:

VSBV,1,2 VPLOT









### 3H.实体建模: 输入/修正



5. 将体分解为对称的模型:

5a.旋转工作平面:

- Utility Menu > Workplane > Offset WP by increments ...
  - 绕 X轴旋转90度,然后选择[OK]
- \_ 或用命令:

WPRO,,90,

5b. 用工作平面切分体:

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Volu by WrkPlane
  - 选择连杆,然后按[OK]
- Utility Menu > Plot > Volumes
- \_ 或用命令:

VSBW,2

VPLOT

5c. 删除连杆的下半部分:

- Main Menu > Preprocessor > Modeling > Delete > Volume and Below
  - 选择体的下部,然后按[OK]
- Utility Menu > Plot > Volumes
- 或用命令:

```
VDELE,3, , ,1
```

```
VPLOT
```











### 3H.实体建模: 输入/修正 **连杆**





- Main Menu > Preprocessor > Modeling > Create > Volumes > Arbitrary > By Areas
  - 拾取组成倒角的体的五个面,然后选择[OK]
- 或用命令:

VA,8,10,12,13,15



3d.将倒角与连杆相加:

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Volumes
  - 拾取[Pick All]
- 或用命令:
  - VADD,all
  - VPLOT

### 3H.实体建模: 输入/修正





3e. 面相加:

- Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas
  - 拾取连杆的平滑部分 /倒角的前表面 (面 14 和 13), 按 [Apply]
  - 拾取连杆的平滑部分 /倒角的后表面 (面 12 和 3), 然后选择[OK]
- 或用命令:

AADD,14,13 AADD,12,3

VPLOT



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### 3H.实体建模: 输入/修正





Training Manual

7.存储数据库,退出ANSYS:

- 在工具条中拾取 "SAVE\_DB"
- 在工具条中拾取 "QUIT"
  - 选择 "Quit No Save!"
  - 按[OK]
- 或用命令:

SAVE

FINISH

/EXIT,NOSAVE

