PART DESIGN

Computational Design Laboratory
Department of Automotive Engineering
Hanyang University, Seoul, Korea

한양대학교 HANYANG UNIVERSITY

CDL Computational Design Lab

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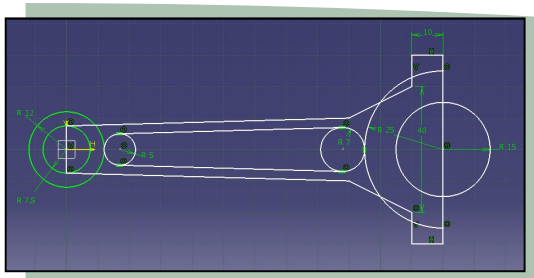
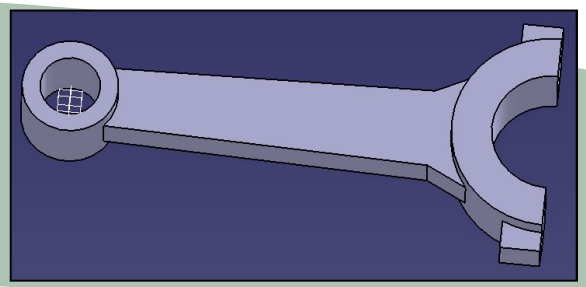
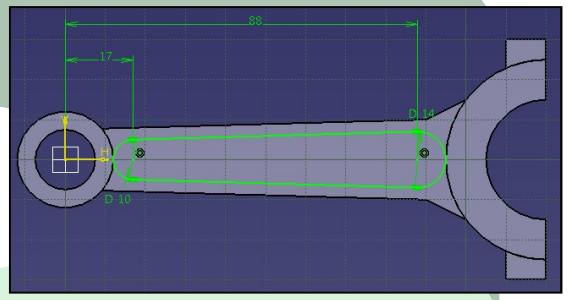
PART DESIGN 순서



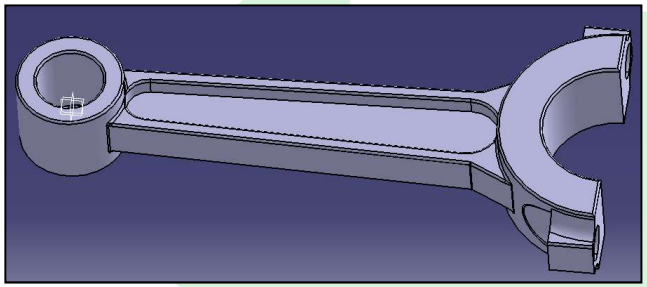
스케치



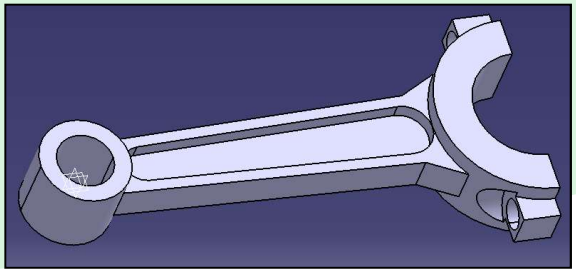
기본 3D 모델 생성



세부 작업을
위한 스케치



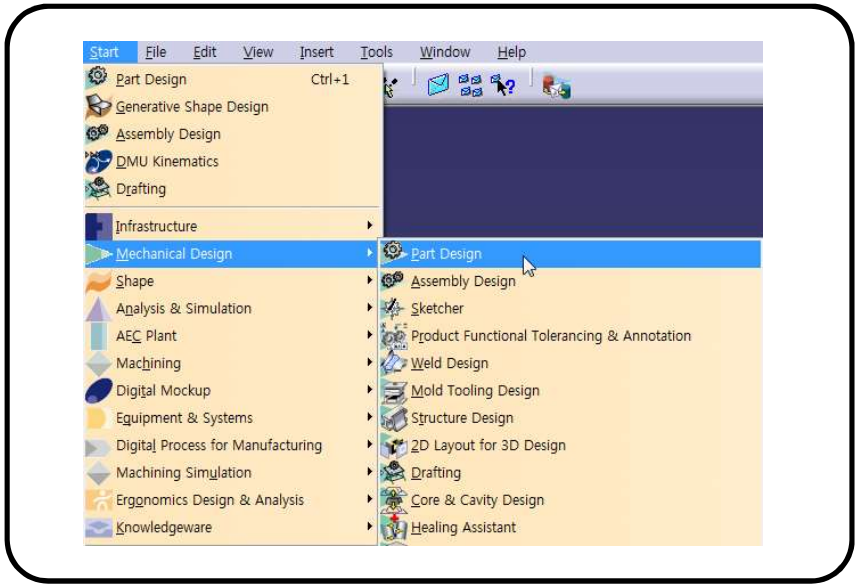
필렛 등의 세부
작업으로 완성



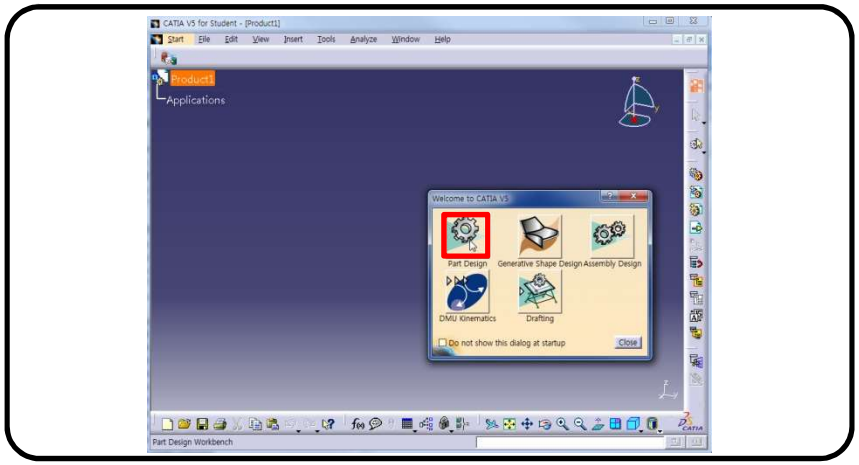
스케치 기반 2차 모델링

PART DESIGN 시작하기

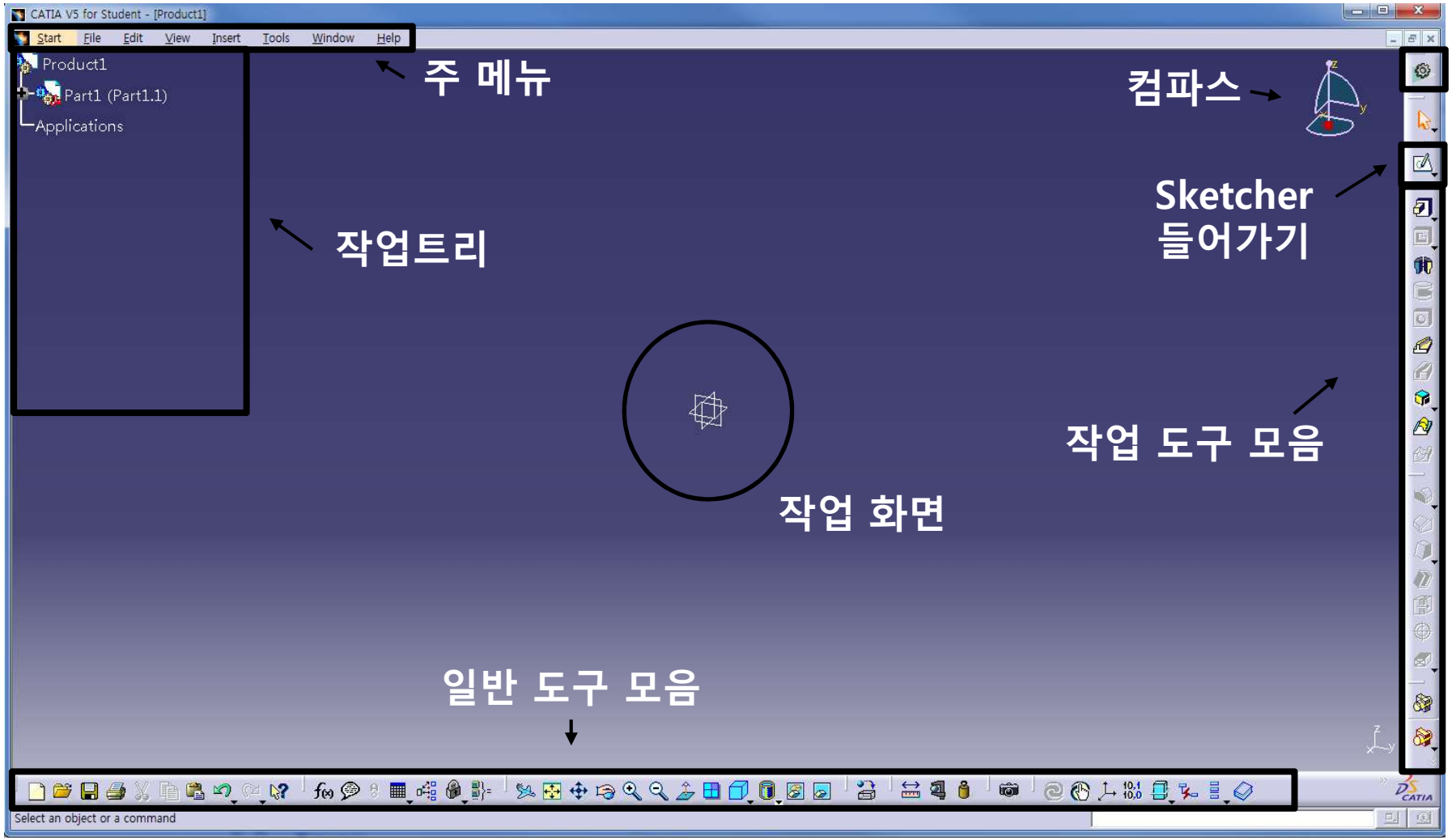
Start 메뉴에서 Mechanical Design → Part Design 선택



혹은, CATIA 실행화면에서 선택 (Start menu 설정 시)



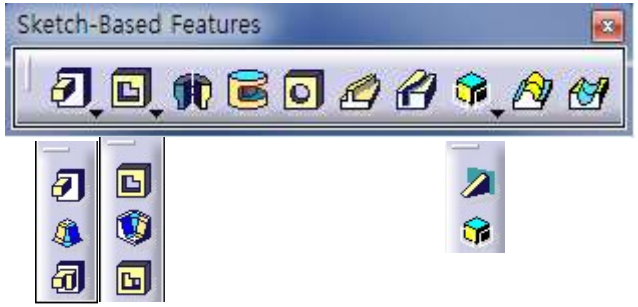
PART DESIGN 작업화면



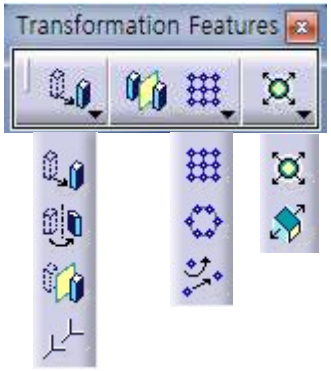
PART DESIGN 툴바 종류

Reference Elements

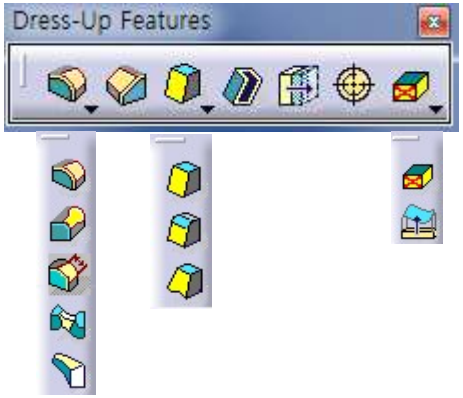
Sketch-Based Features



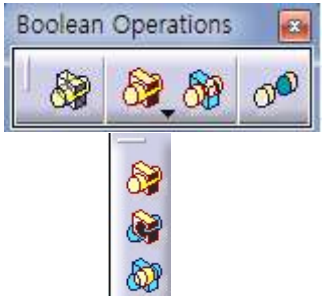
Transformation Features



Dress-Up Features



Boolean Operations



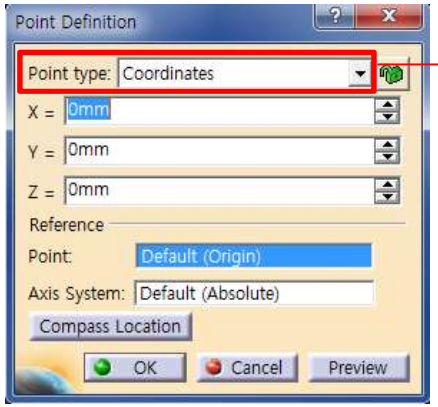
CONTENTS

PART DESIGN TOOLS

Reference Elements

Point

- 작업 화면상에 Point를 생성



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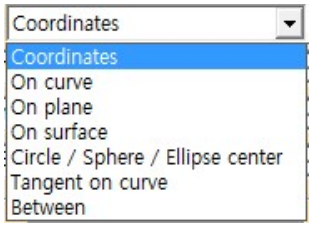

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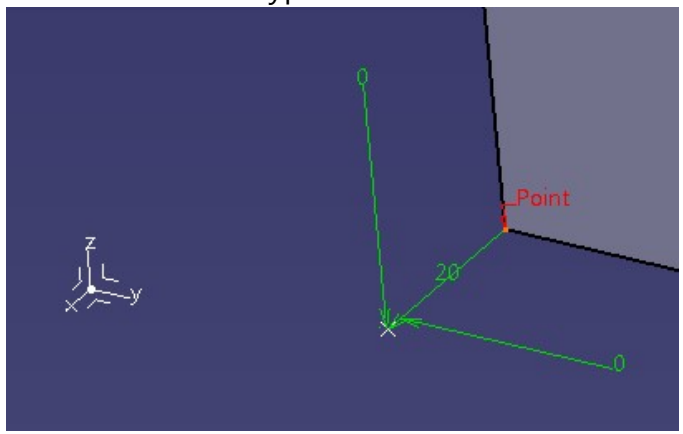

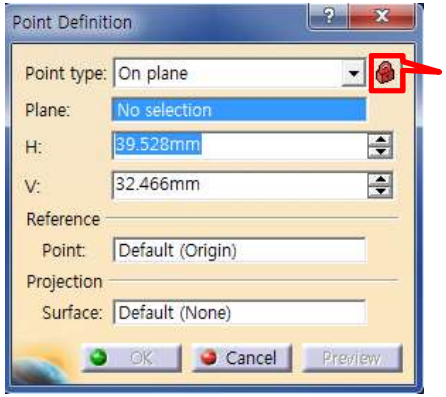

<기능 잠금>
: 자동으로 메뉴가 바뀌지 않도록 고정

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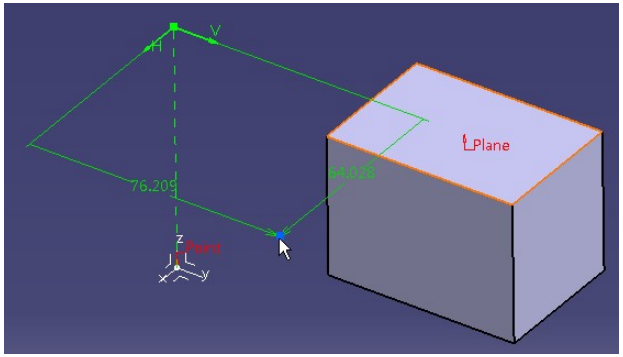
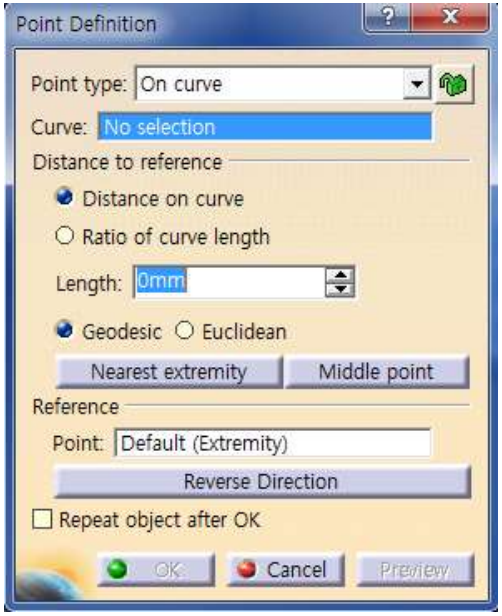


PART DESIGN TOOLS

Reference Elements

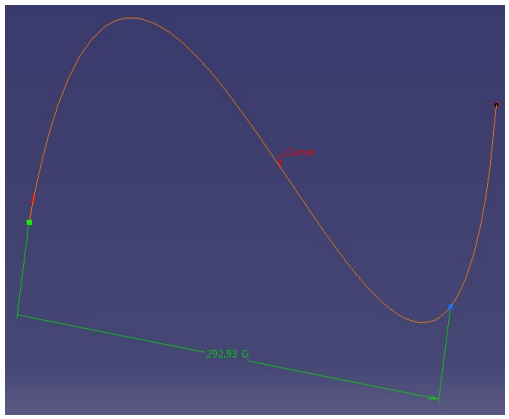
- Point type: On curve
  - Curve위에 점을 생성



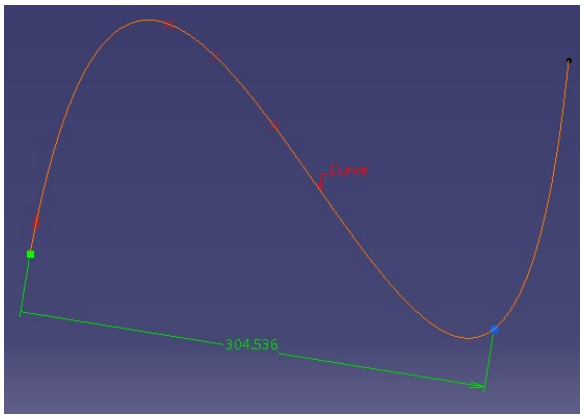
Geodesic: 곡선을 포함한 거리

Euclidean: 직선 거리

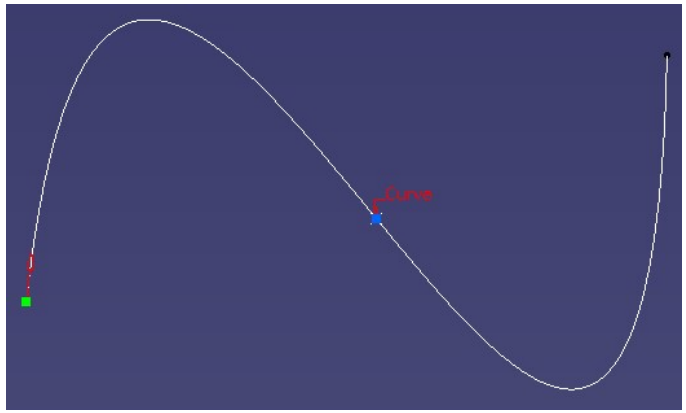
<Geodesic>



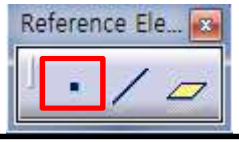
<Euclidean>



<Middle point(Geodesic)>

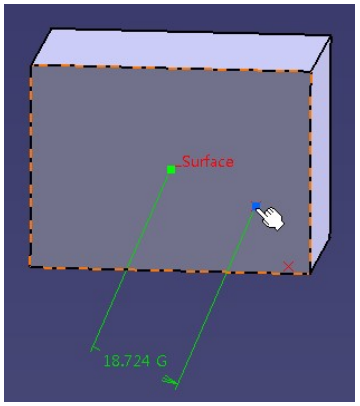


PART DESIGN TOOLS

Reference Elements 

- Point type: On surface
  - 면이나 평면상에 점을 생성(Geodesic)

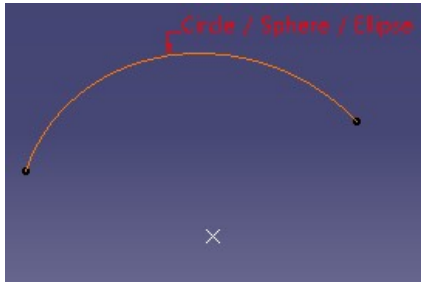
<Surface>



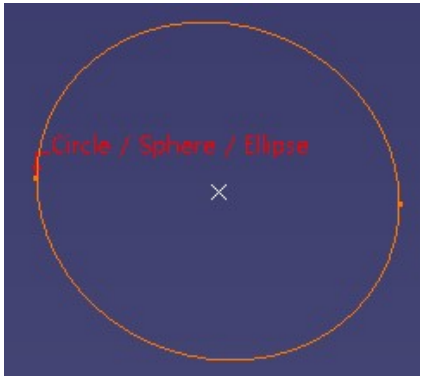
<Plane>

- Point type: Circle/Sphere/Ellipse center
  - Arc, Circle 등의 중심에 점 생성

<Arc>

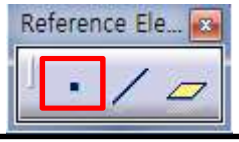


<Circle>

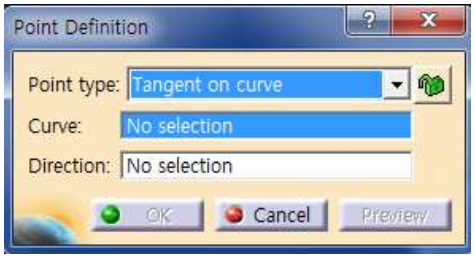


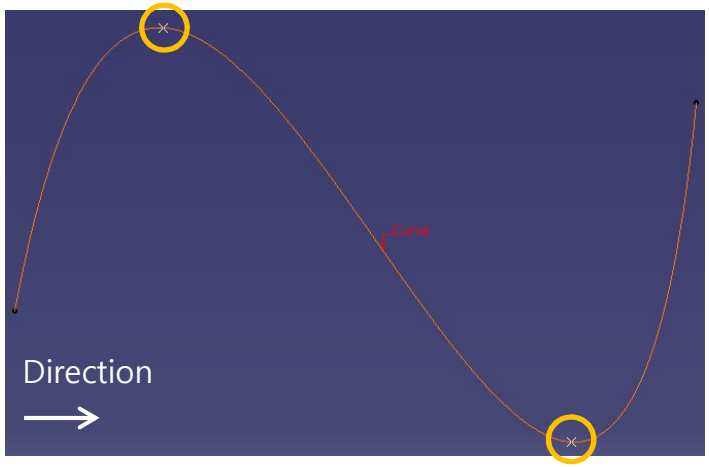
PART DESIGN TOOLS

Reference Elements

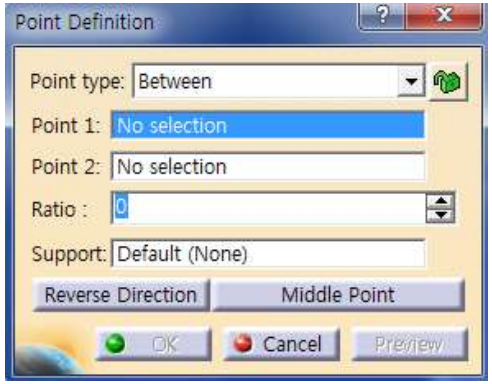


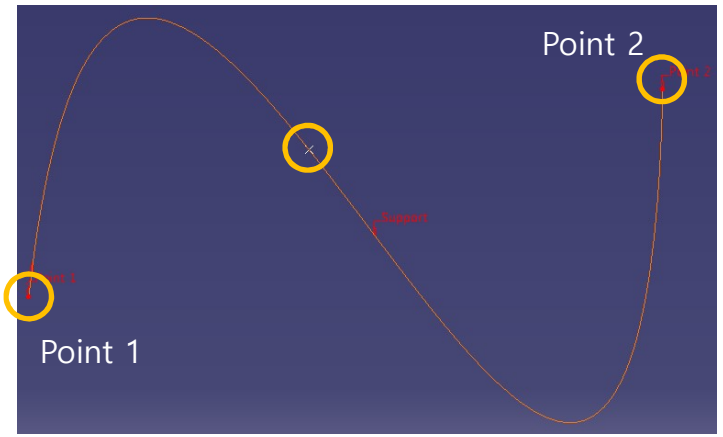
- Tangent on curve
  - 선택한 방향의 기울기를 갖는 위치에 점 생성



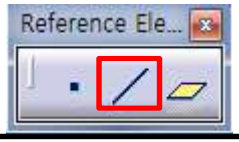


- Between
  - 두 점 사이에 새로운 점을 생성



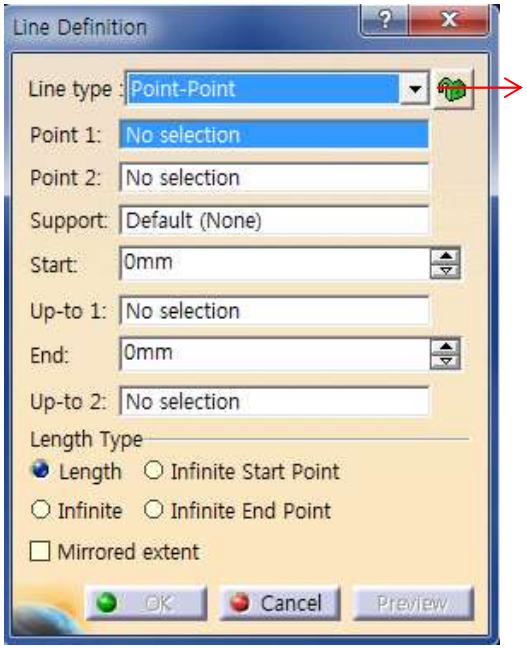


PART DESIGN TOOLS

Reference Elements 

Line 

- 작업 화면상에 Line를 생성



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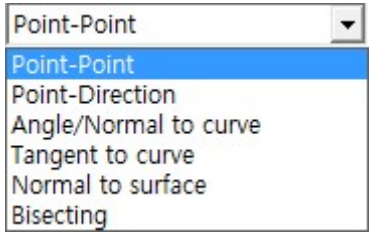

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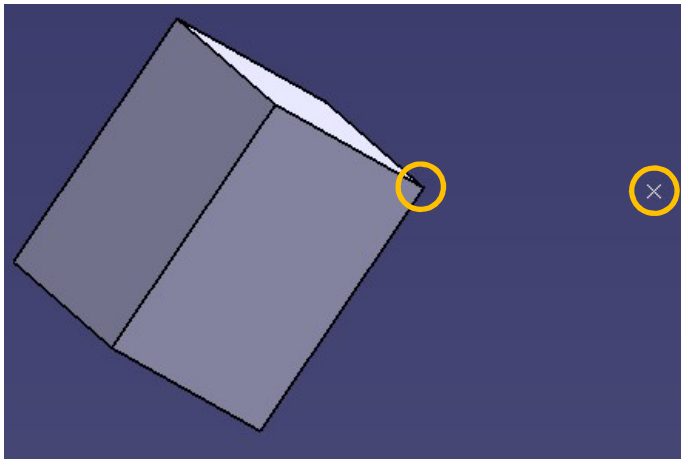
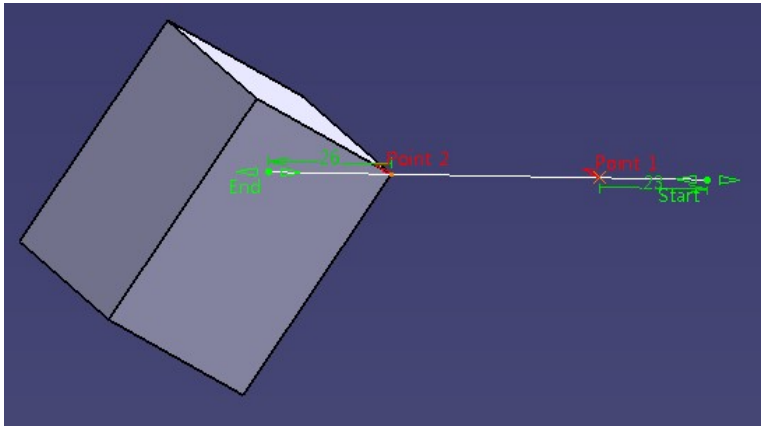
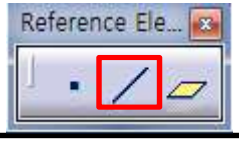


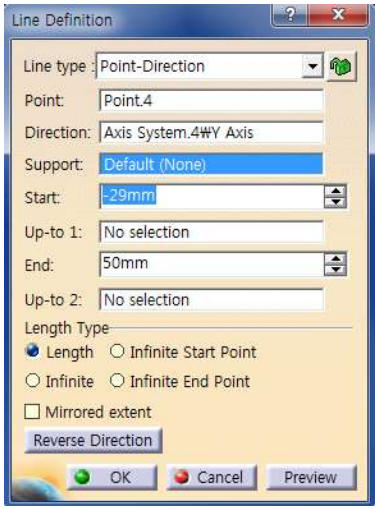


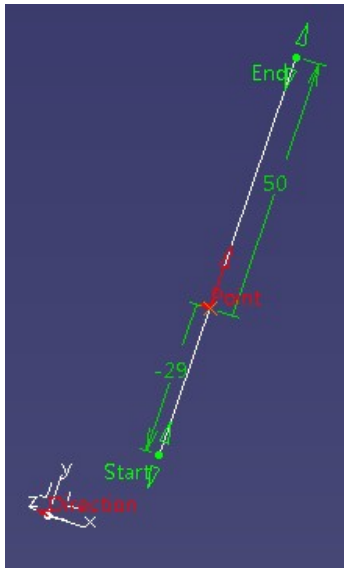
PART DESIGN TOOLS

Reference Elements

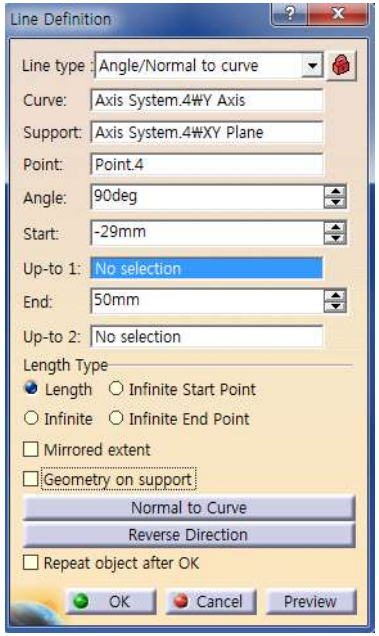


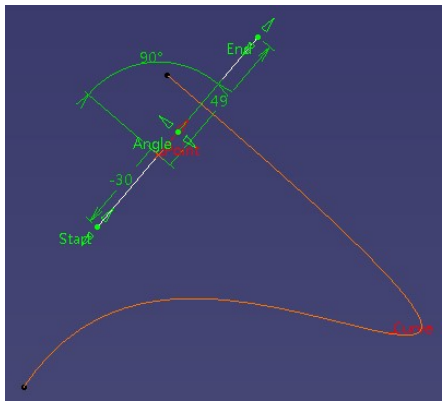
- Point-Direction
  - 점과 방향으로 생성

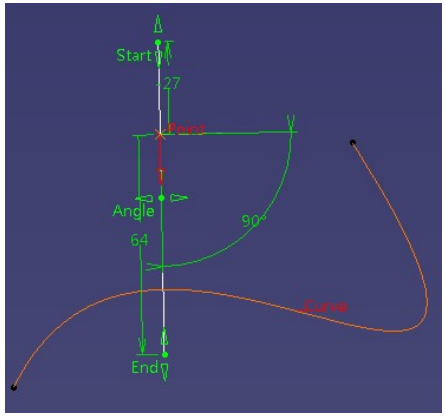




- Angle/Normal to curve
  - curve에서 선택 점에 가장 가까운 위치의 기울기를 기준으로 각도를 갖는 직선 생성

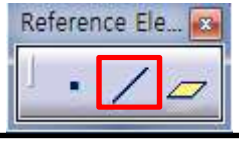




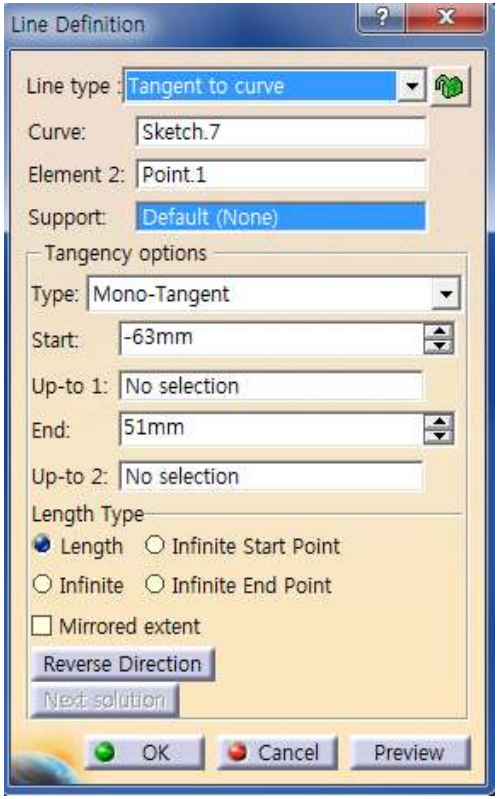


PART DESIGN TOOLS

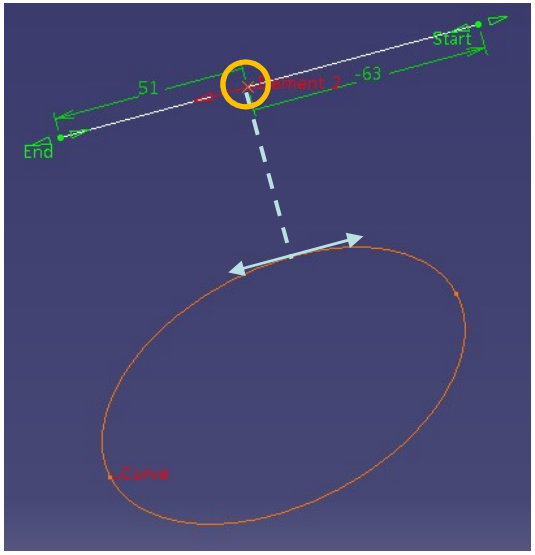
Reference Elements



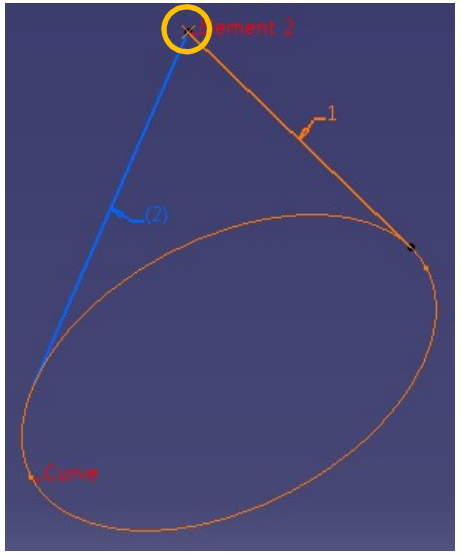
- Tangent to curve
  - Curve에 tangent한 직선 생성



< Mono-Tangent>



<BiTangent>

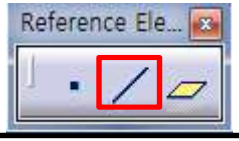


Mono-Tangent: 선택 요소에서 curve에 가장 가까운 점의 접선 방향

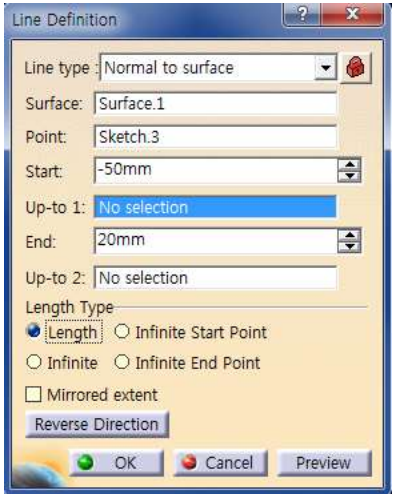
BiTangent: 선택 요소에서 곡선에 접하는 방향

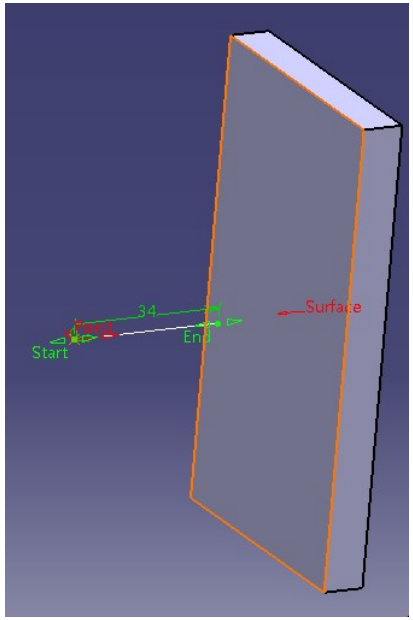
PART DESIGN TOOLS

Reference Elements

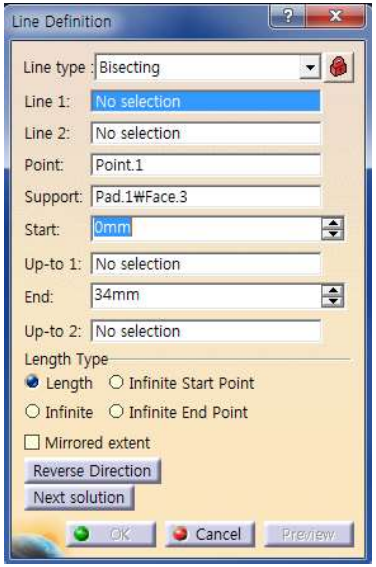


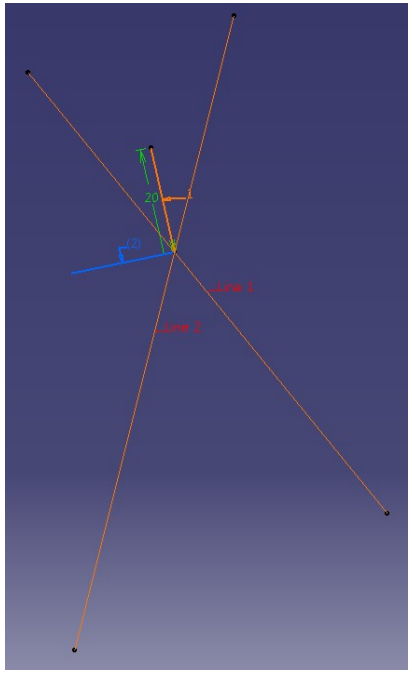
- Normal to surface
  - 점에서 면에 수직한 직선 생성





- Bisecting
  - 교차하는 선을 이등분하는 선 생성





PART DESIGN TOOLS

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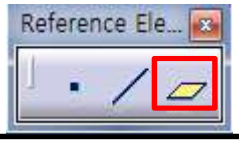
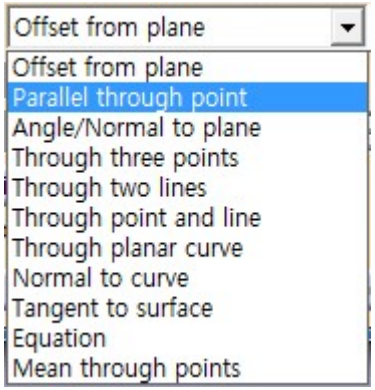


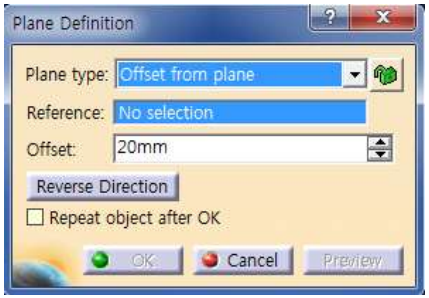
Plane

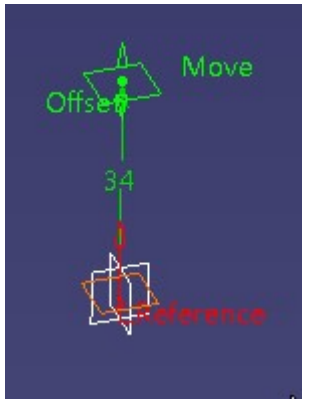
- 사용자가 원하는 위치에 새로운 평면 생성

<Plane type>

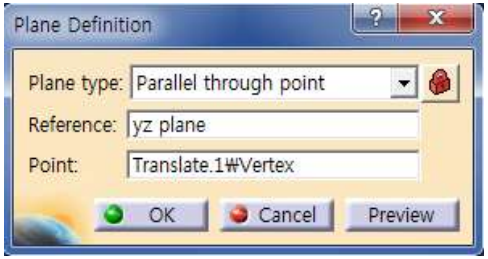


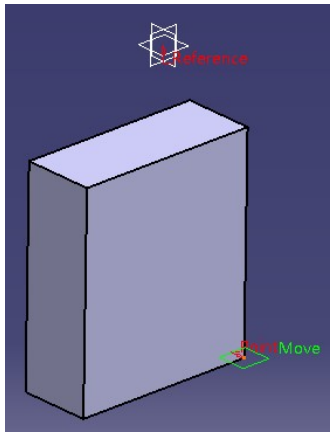
- Offset from plane
  - Reference에서 offset 만큼의 위치에 생성





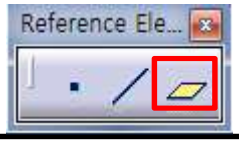
- Parallel through point
  - Reference로 선택한 평면과 평행한 평면을 선택한 점의 위치에 생성



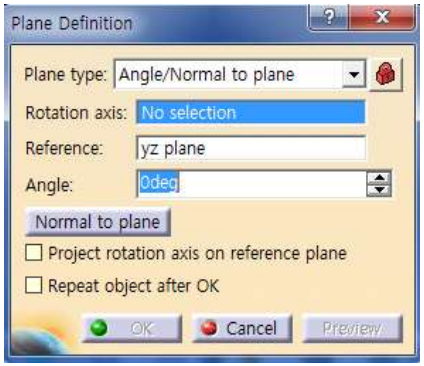


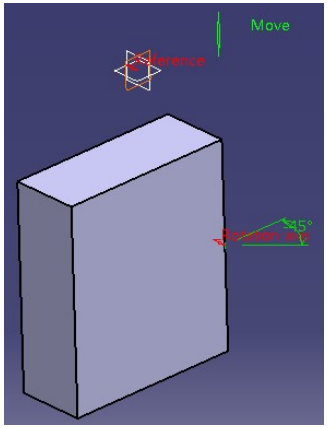
PART DESIGN TOOLS

Reference Elements

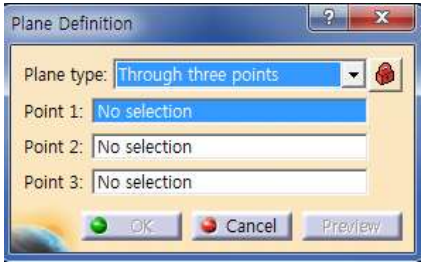


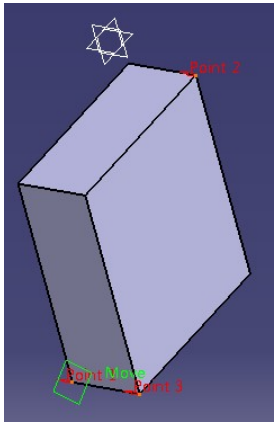
- Angle/Normal to plane
  - 축을 기준으로 회전한 평면



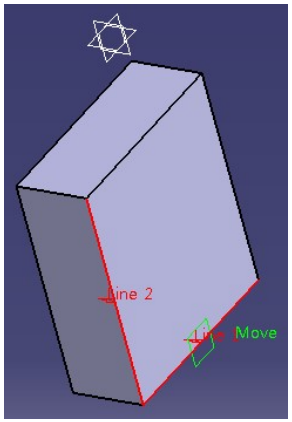


- Through three points
  - 세 점으로 이루어진 평면

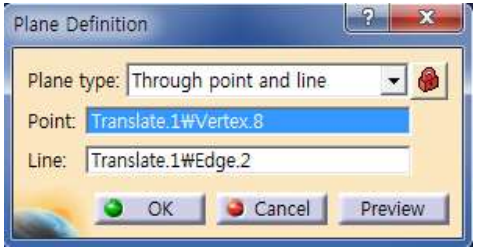


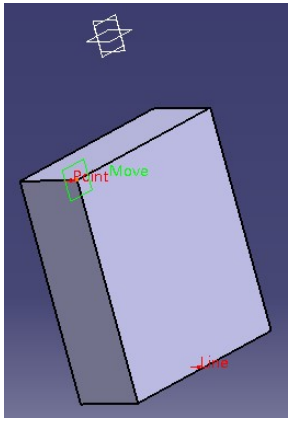


- Through two lines
  - 두 직선으로 이루어진 평면



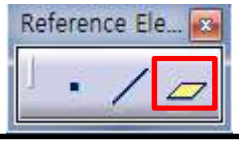
- Through point and line
  - 점과 직선으로 이루어진 평면



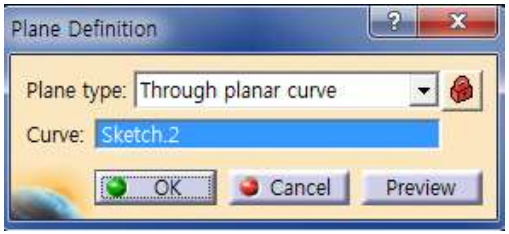


PART DESIGN TOOLS

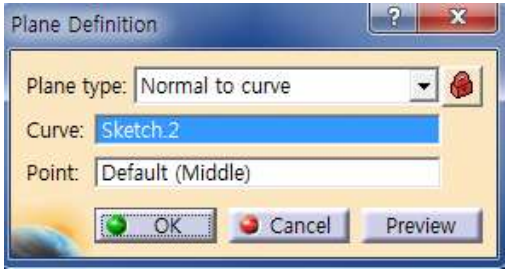
Reference Elements

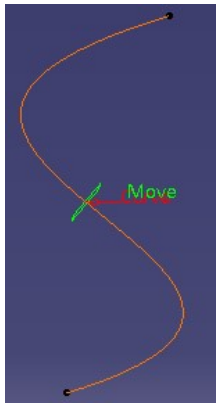


- Through planar curve
  - Curve의 평면

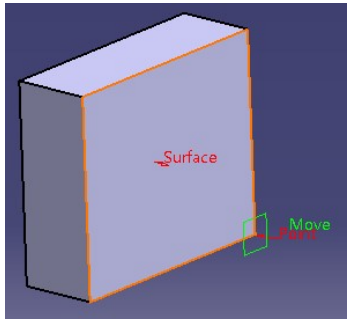


- Normal to curve
  - Curve에 수직한 평면

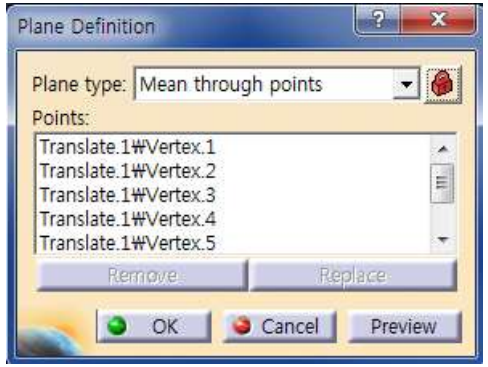


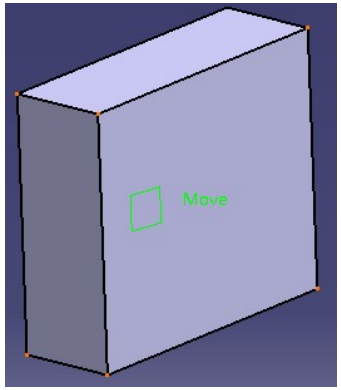


- Tangent to surface
  - 면에 tangent한 평면



- Mean through points
  - 여러 점의 평균 평면 (최소 3개)





CONTENTS

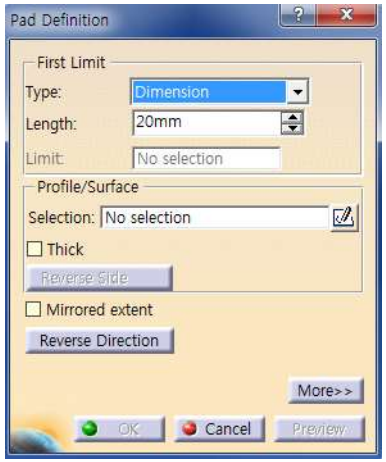
- ✓ 시작하기
- ✓ Reference Elements
- ✓ **Sketch-Based Features**
- ✓ Dress-Up Features
- ✓ Transformation Features
- ✓ Boolean Operation

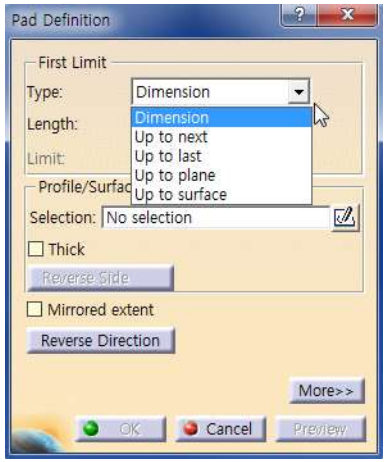
PART DESIGN TOOLS

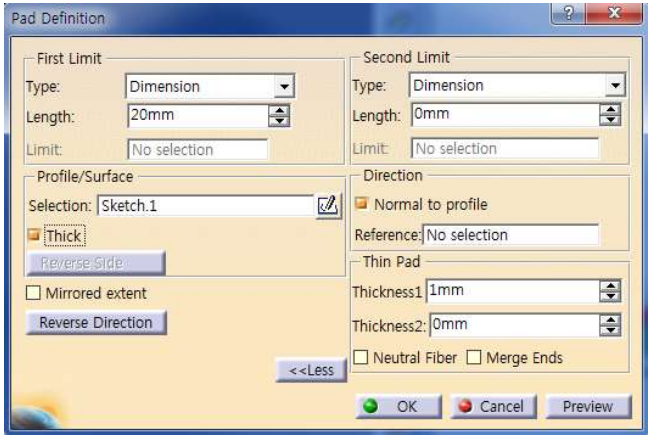
Sketch-Based Features

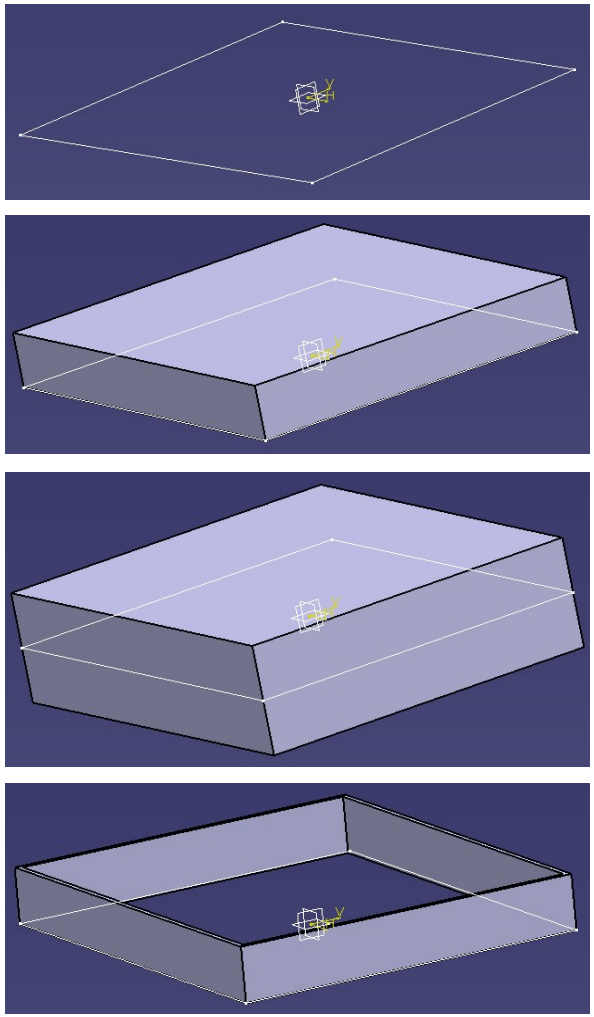
Pad 

- 스케치를 기준으로 3D 모델 생성.







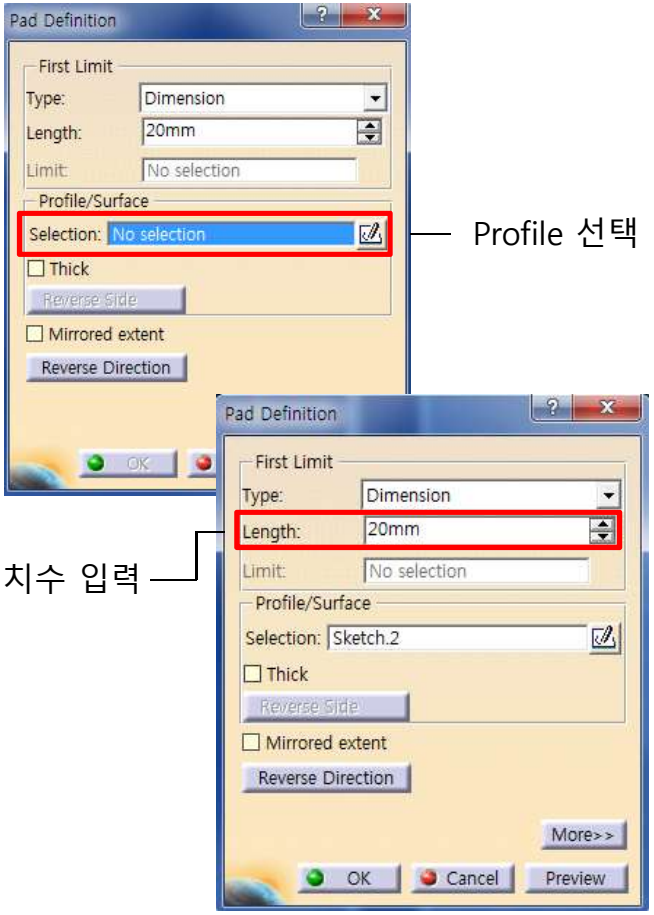


PART DESIGN TOOLS

Sketch-Based Features

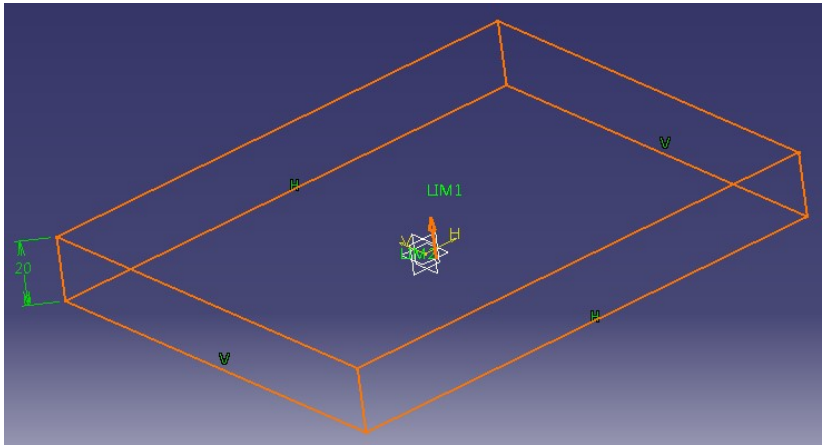
Pad

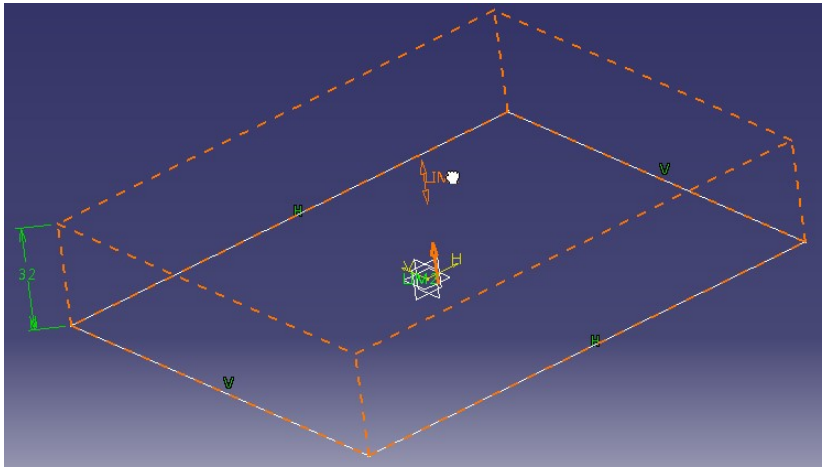
- 기본 기능(Type: Dimension)



Profile 선택

치수 입력





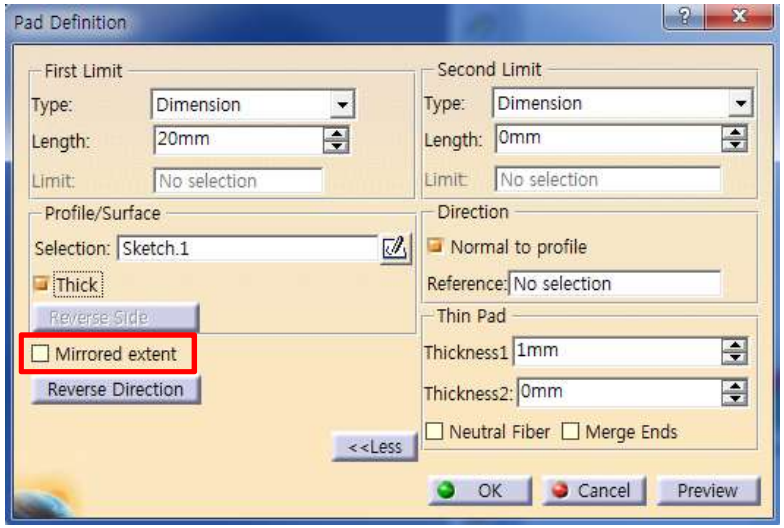
<'LIM'을 클릭하여 높이 조정가능>

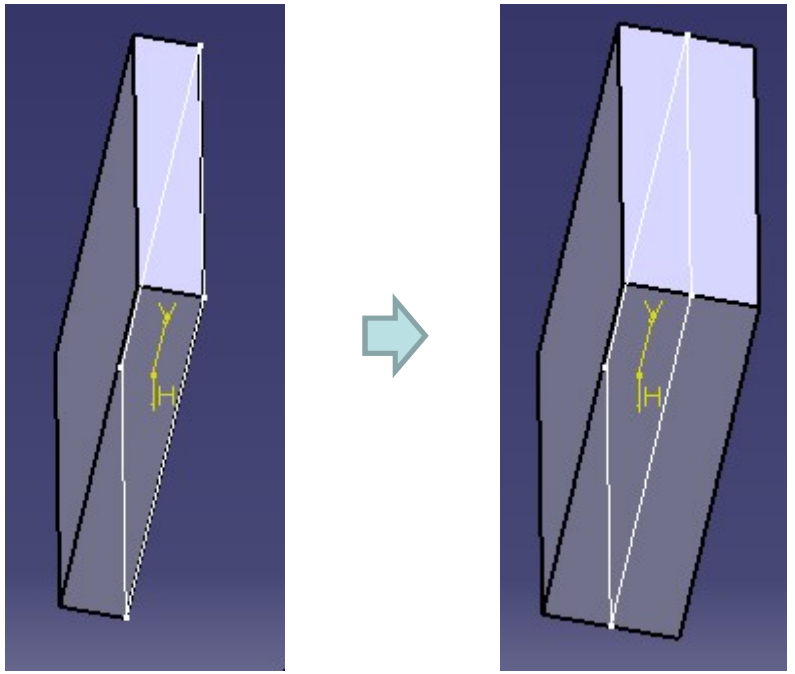
PART DESIGN TOOLS

Sketch-Based Features

Pad 

-Mirrored extent : Profile에 대칭으로 생성



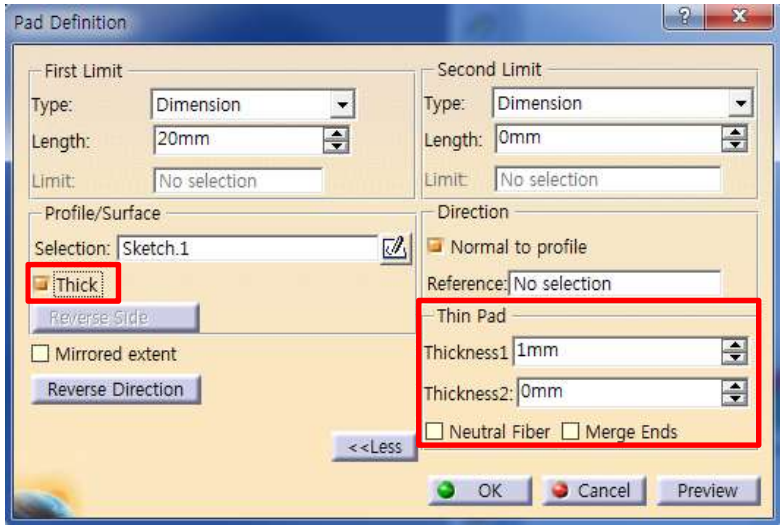


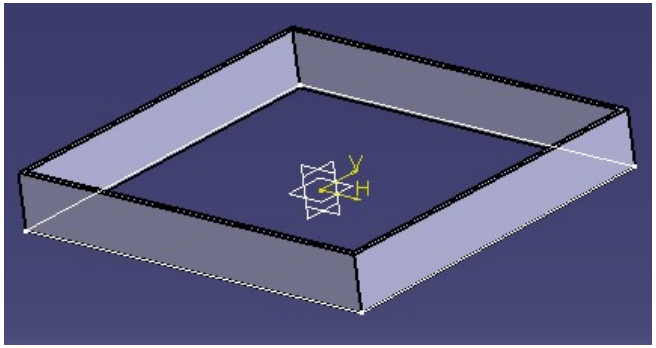
PART DESIGN TOOLS

Sketch-Based Features

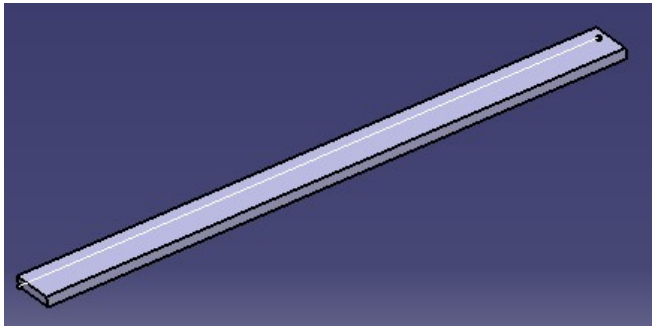
Pad

-Thick : 두께를 갖는 판 생성





<Thick>



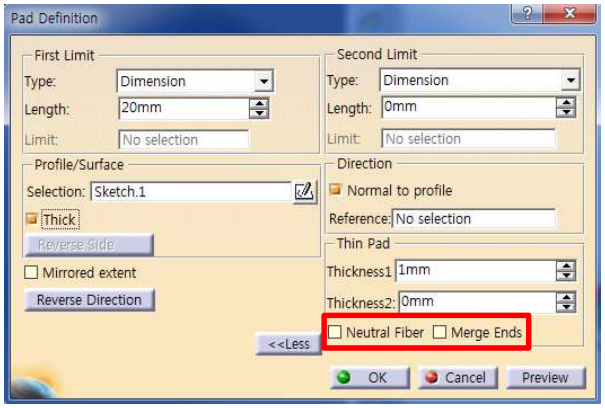
<Open profile>

PART DESIGN TOOLS

Sketch-Based Features

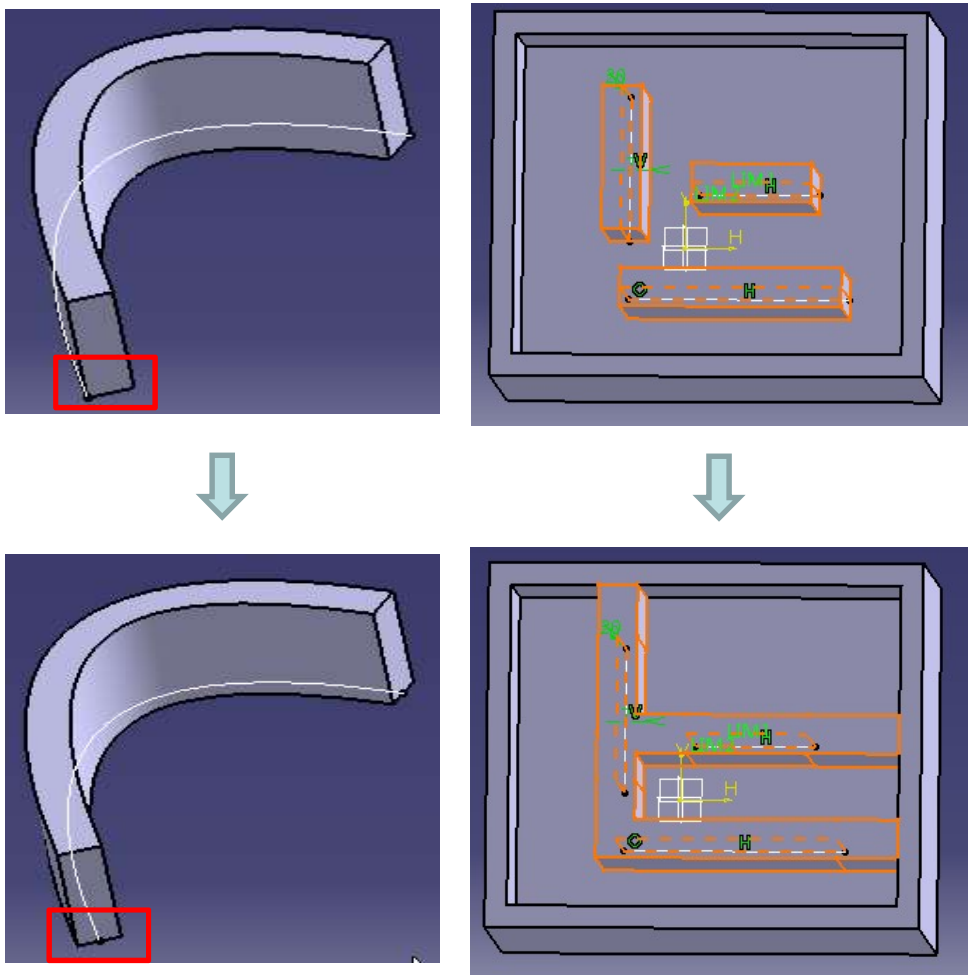
Pad 

-Thick : 두께를 갖는 판 생성



Neutral Fiber: 스케치가 두께의 중심부에 위치하도록 설정

Merge Ends: 스케치의 연장선까지 생성



<Neutral Fiber>

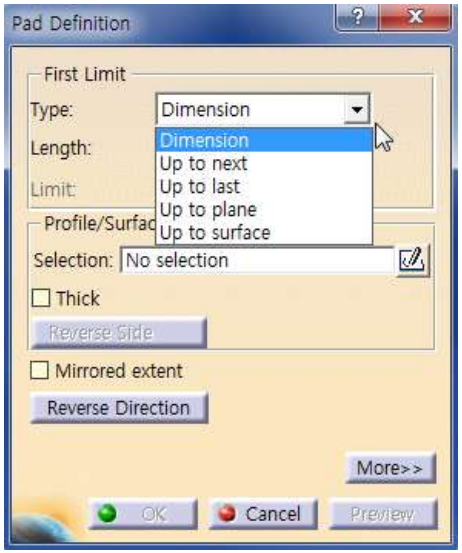
<Merge Ends>

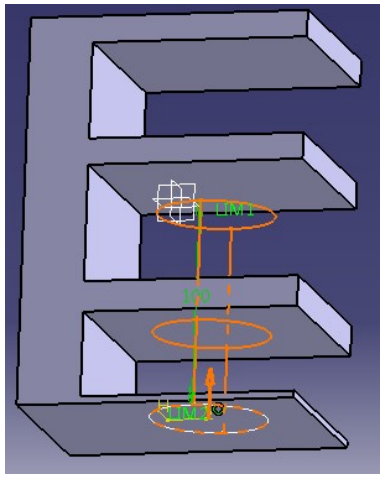
PART DESIGN TOOLS

Sketch-Based Features

Pad 

-Pad type





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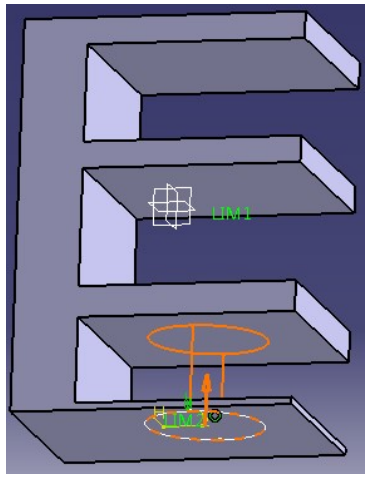

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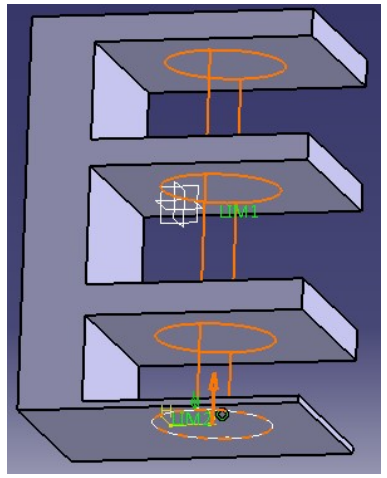

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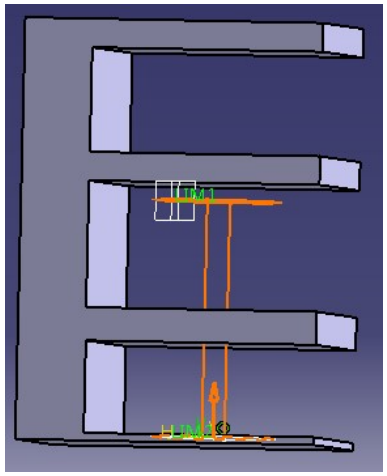
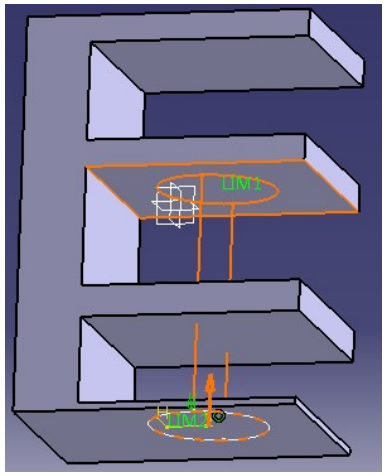


<Up to plane>



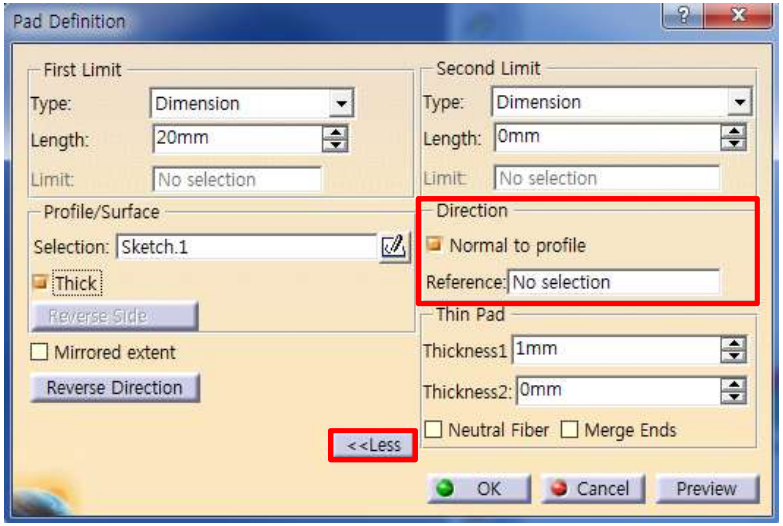
<Up to surface>

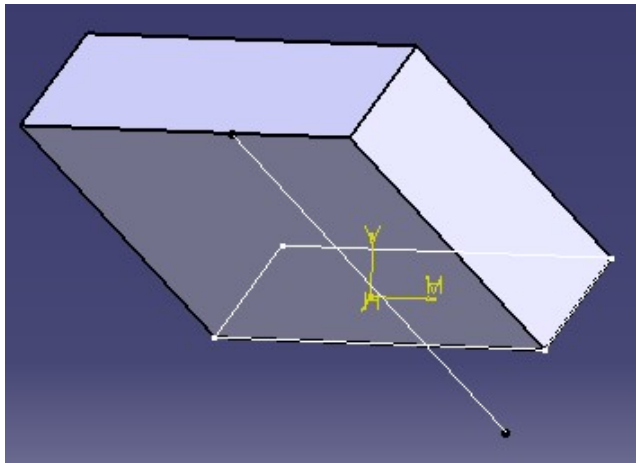
PART DESIGN TOOLS

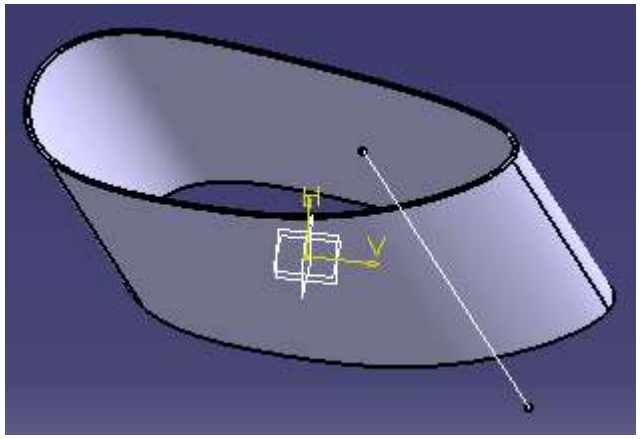
Sketch-Based Features

Pad

-Direction 설정 (직선)





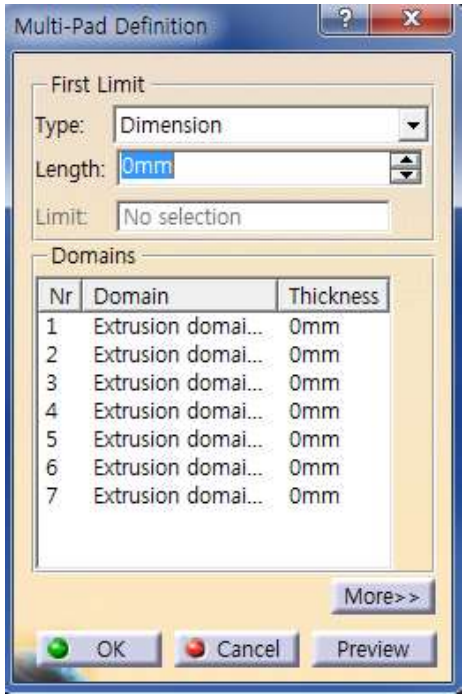


PART DESIGN TOOLS

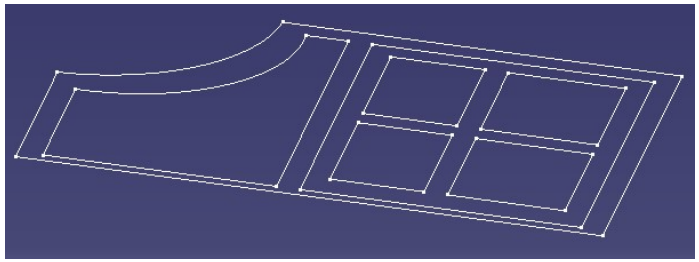
Sketch-Based Features

Multi-Pad

- 여러 개의 스케치를 동시에 Pad



<Profile>



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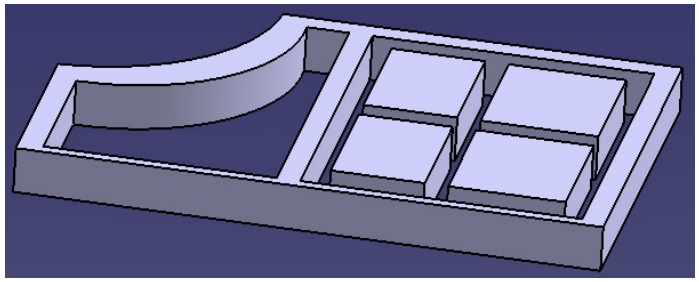
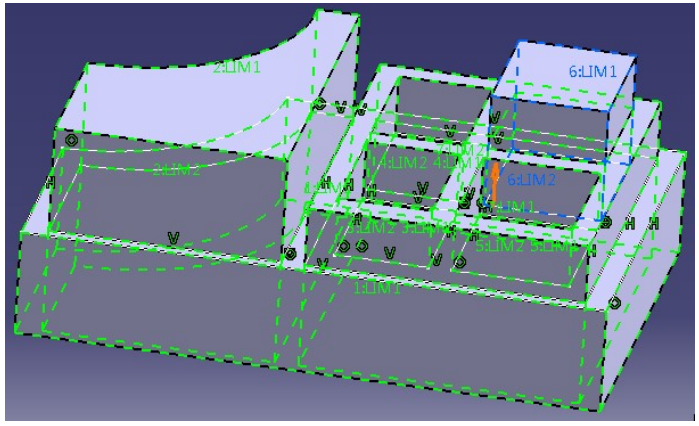


<Multi-Pad>

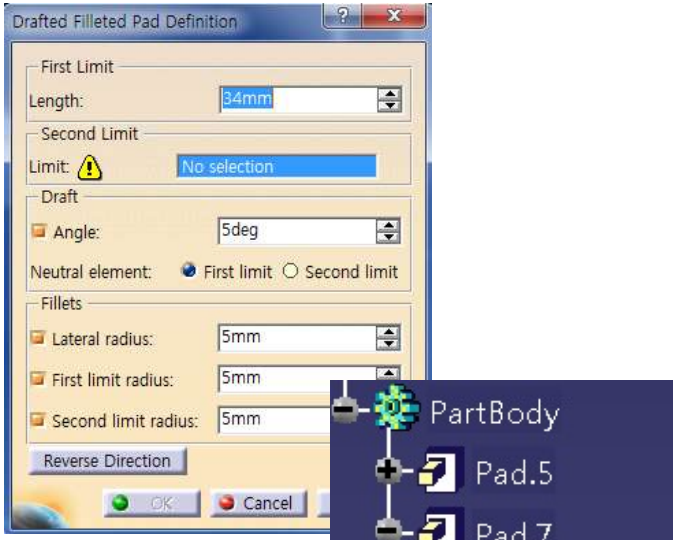


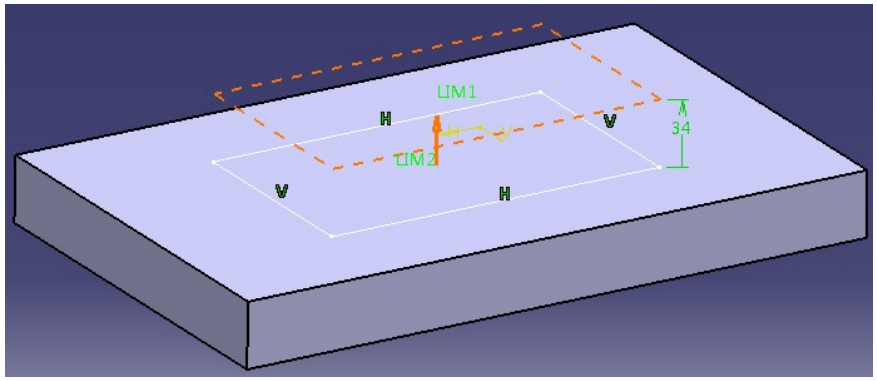
PART DESIGN TOOLS

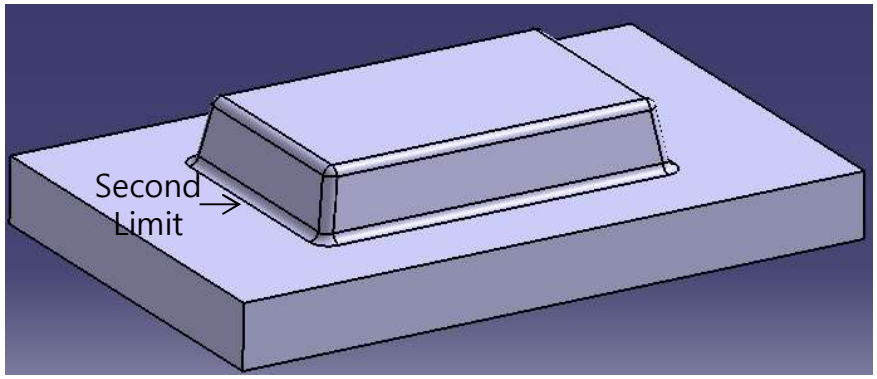
Sketch-Based Features

Drafted Filleted Pad

- Pad와 동시에 Draft와 Fillet 진행





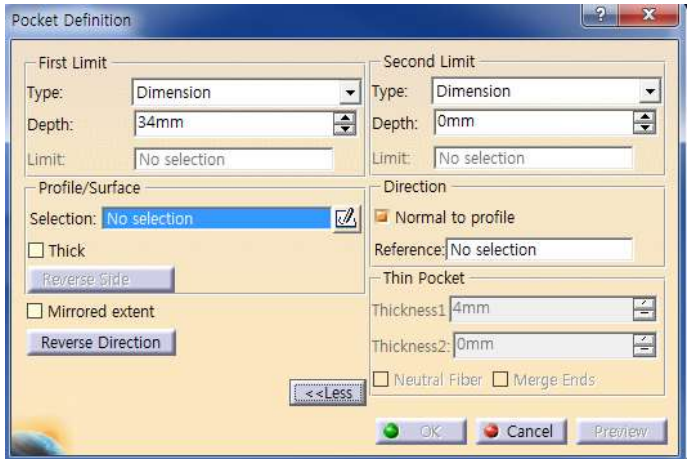


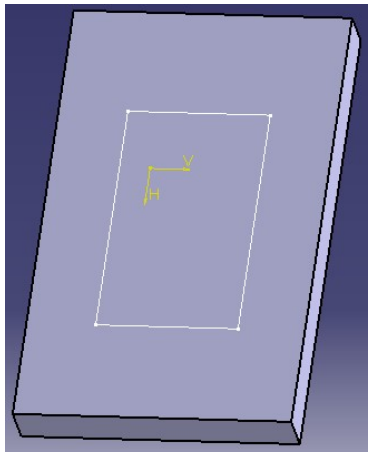
PART DESIGN TOOLS

Sketch-Based Features

Pocket 

- 스케치를 기준으로 기존 모델의 일부 형상제거





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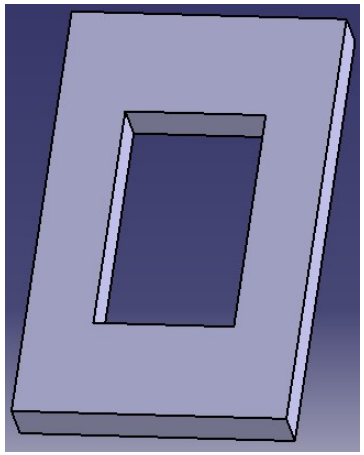

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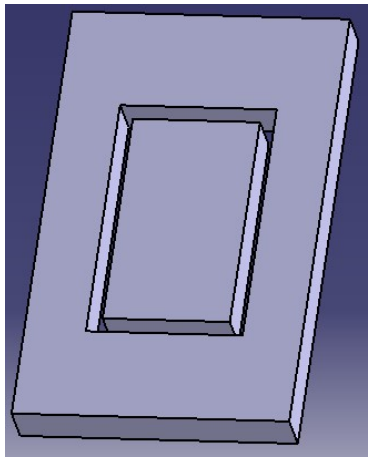
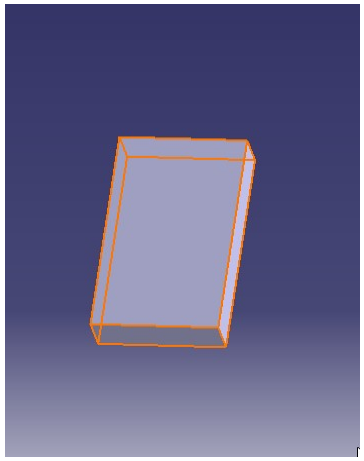


<Thick>

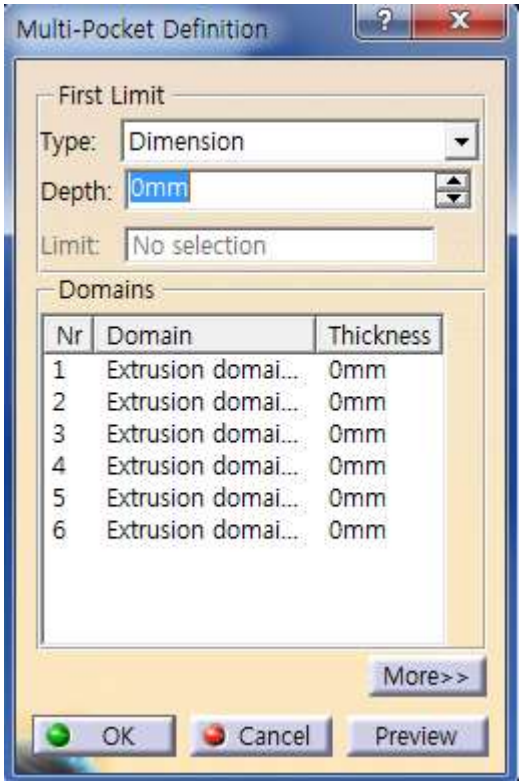


<Reverse Direction>

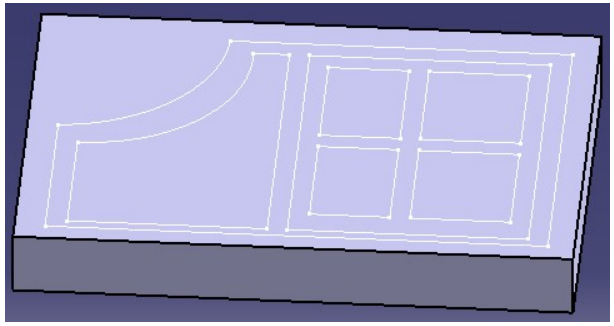
PART DESIGN TOOLS

Sketch-Based Features

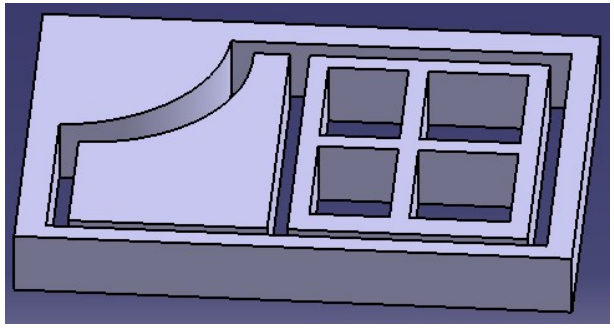
Multi-Pocket
- 여러 개의 스케치를 동시에 Pocket



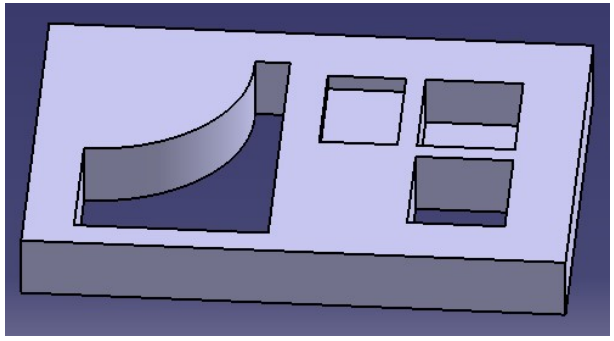
<Profile>



<Pocket>



<Multi-Pocket>

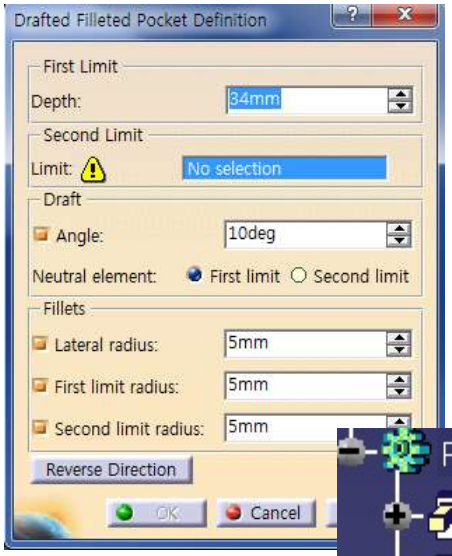


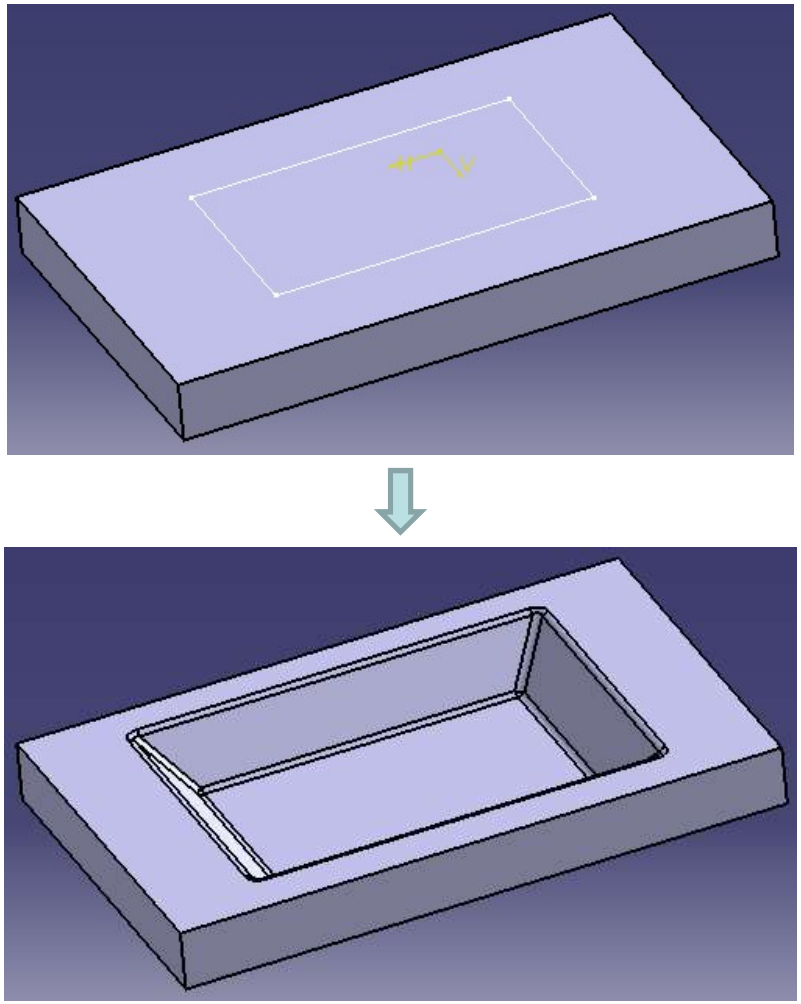
PART DESIGN TOOLS

Sketch-Based Features

Drafted Filleted Pocket

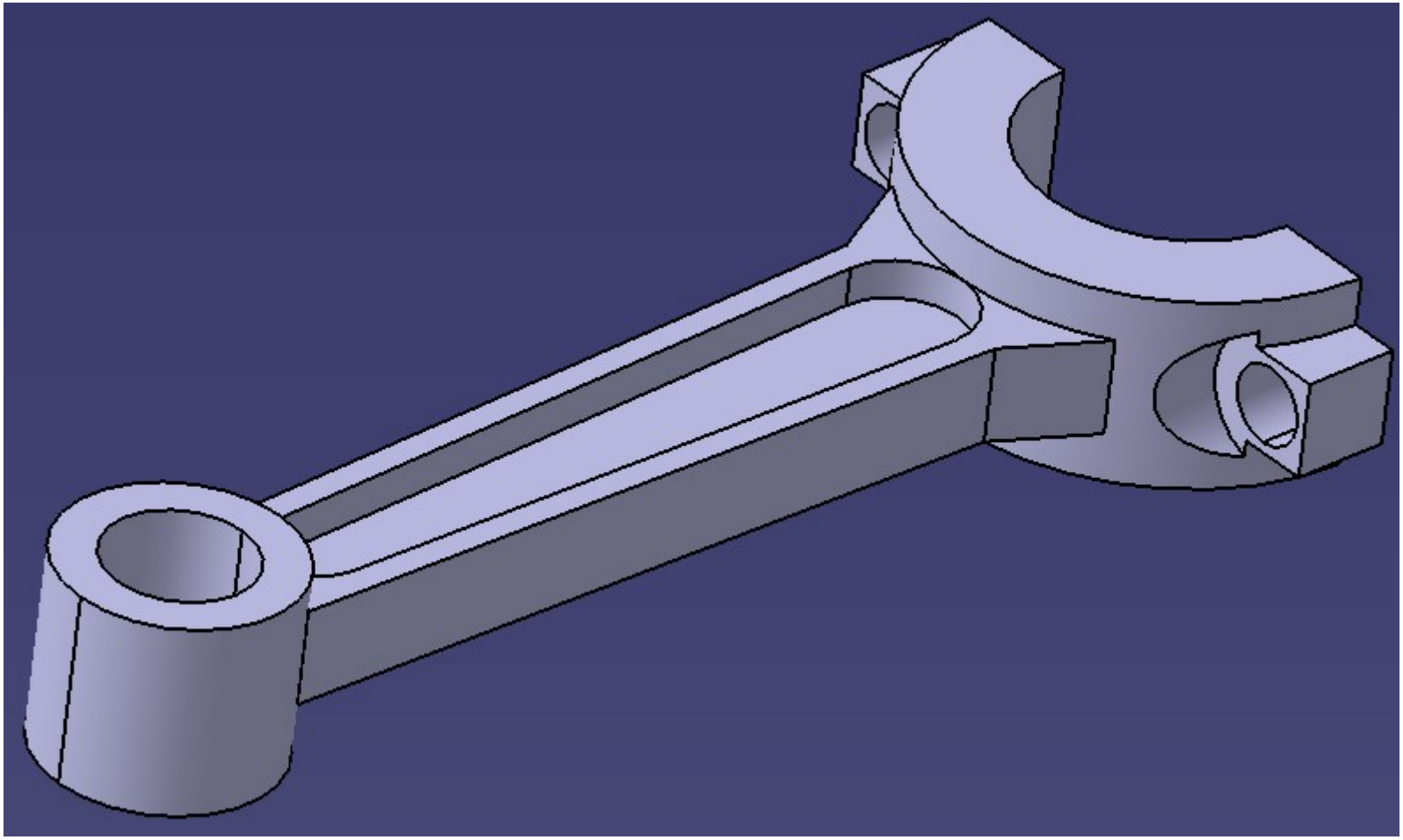
- Pad와 동시에 Draft와 Fillet 진행





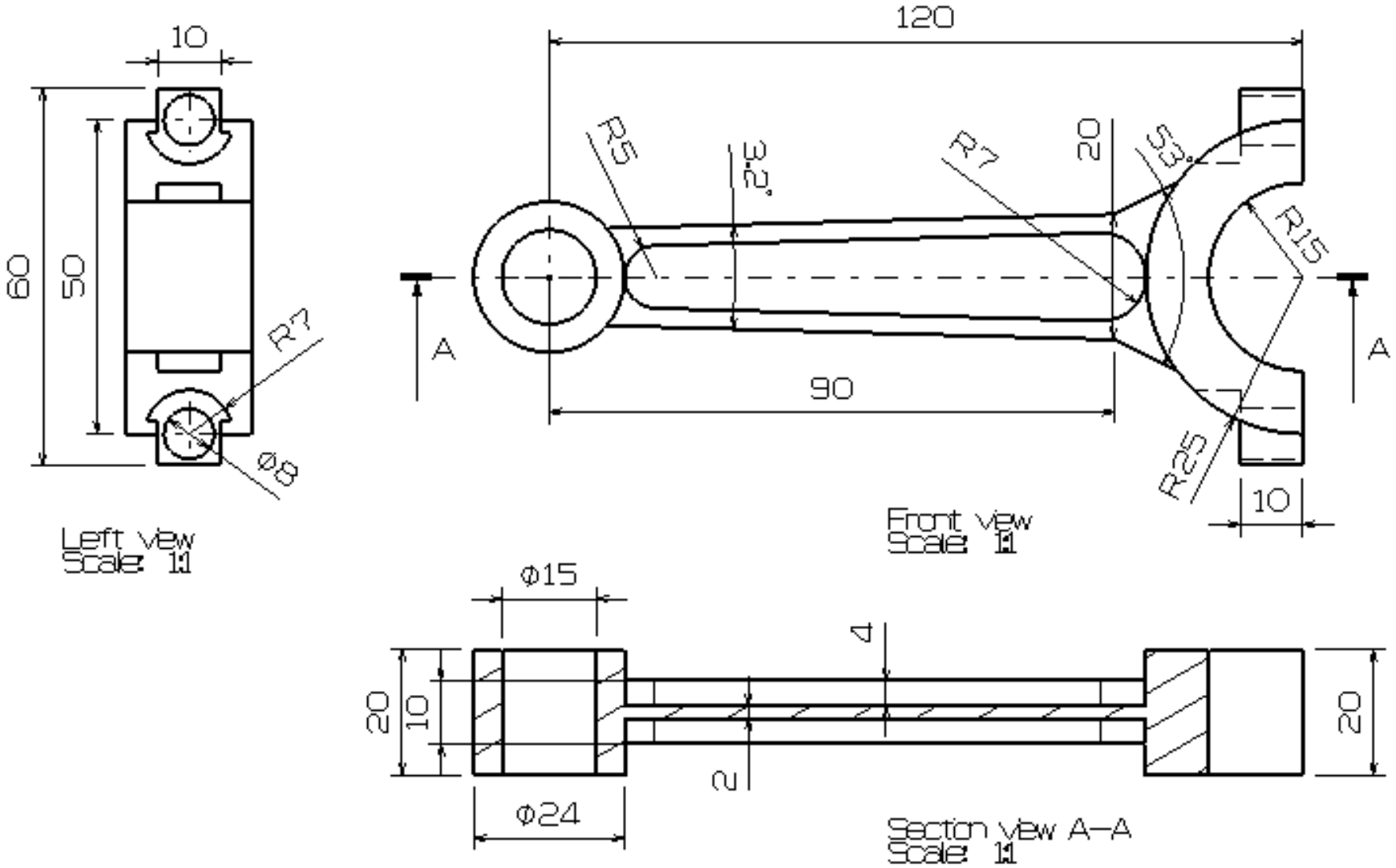
실습 예제

Connecting Rod



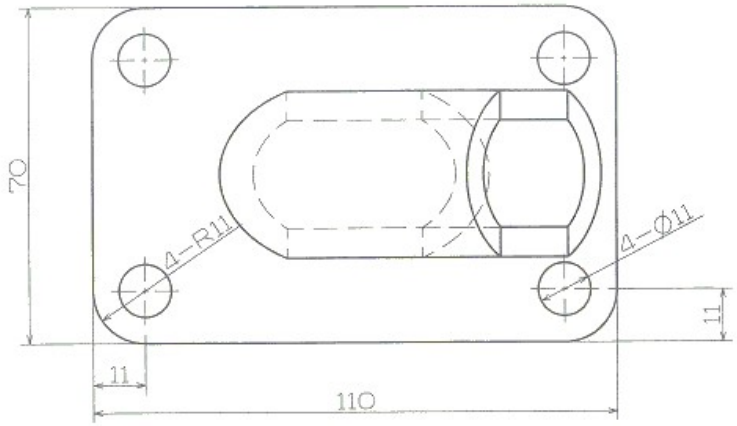
실습 예제

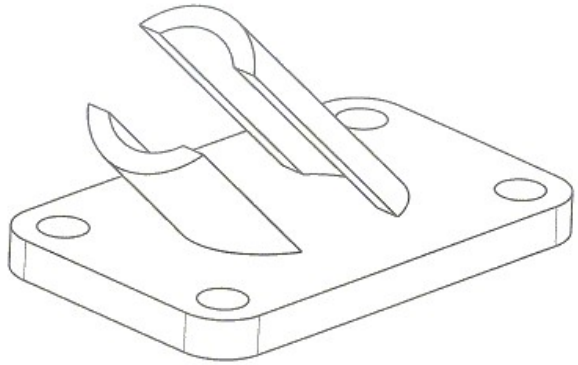
Connecting Rod

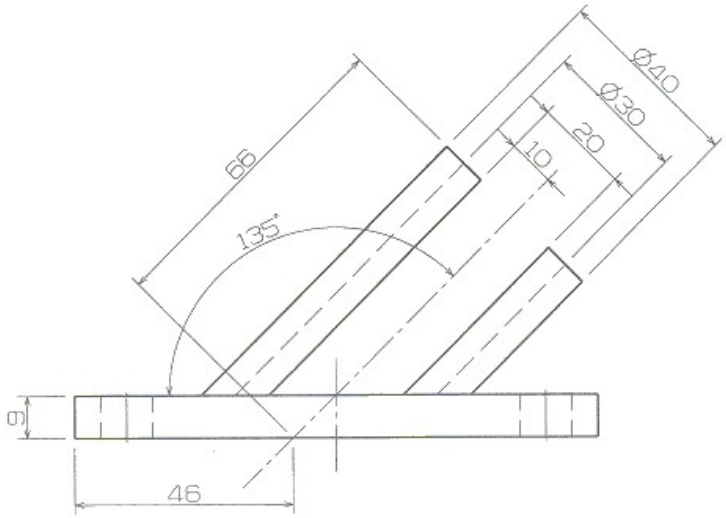


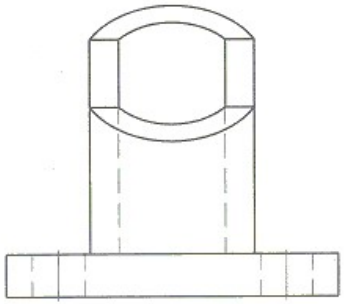
실습 예제

다음 도면을 참고하여 모델링을 진행하시오. (Plane element 이용)



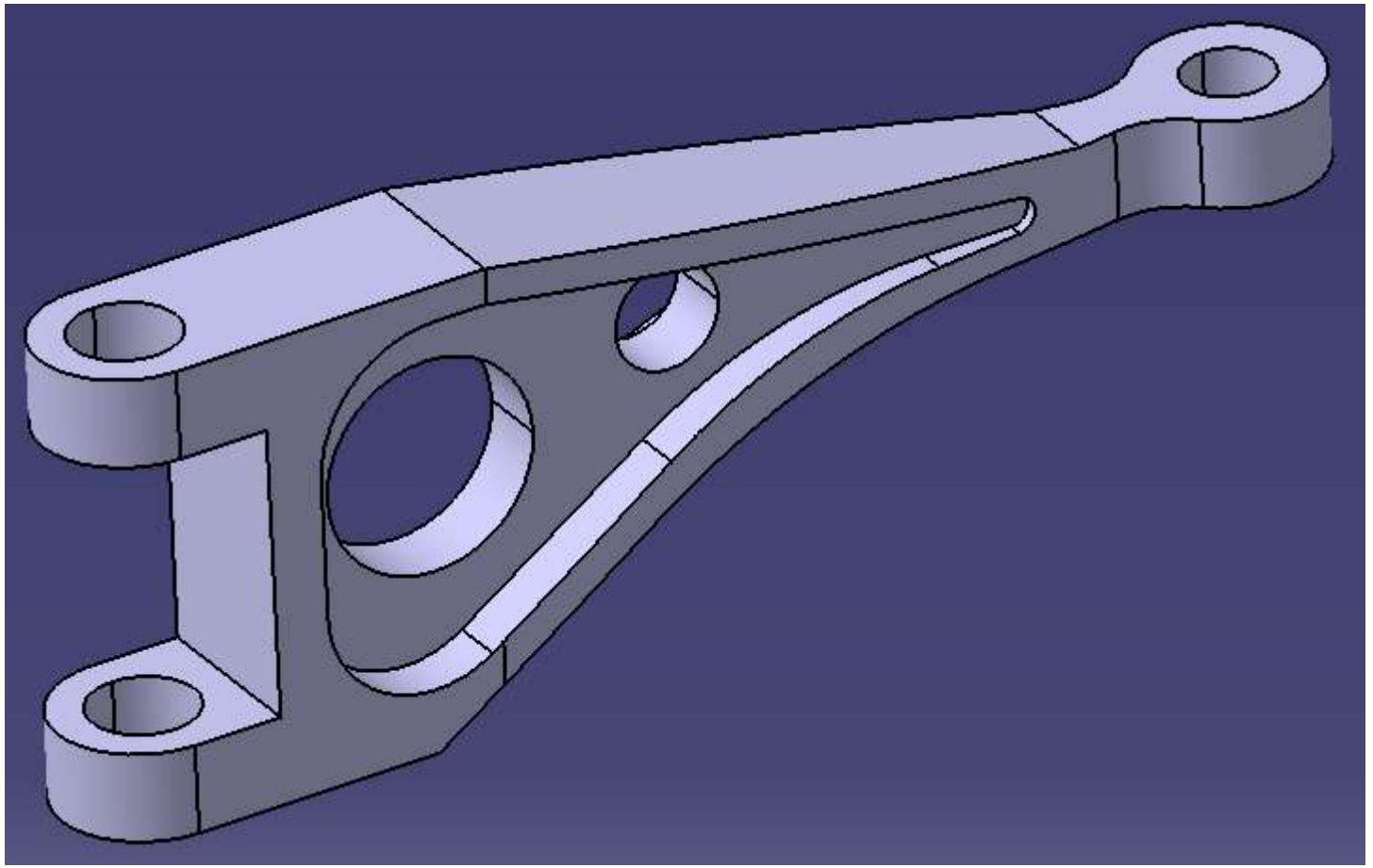






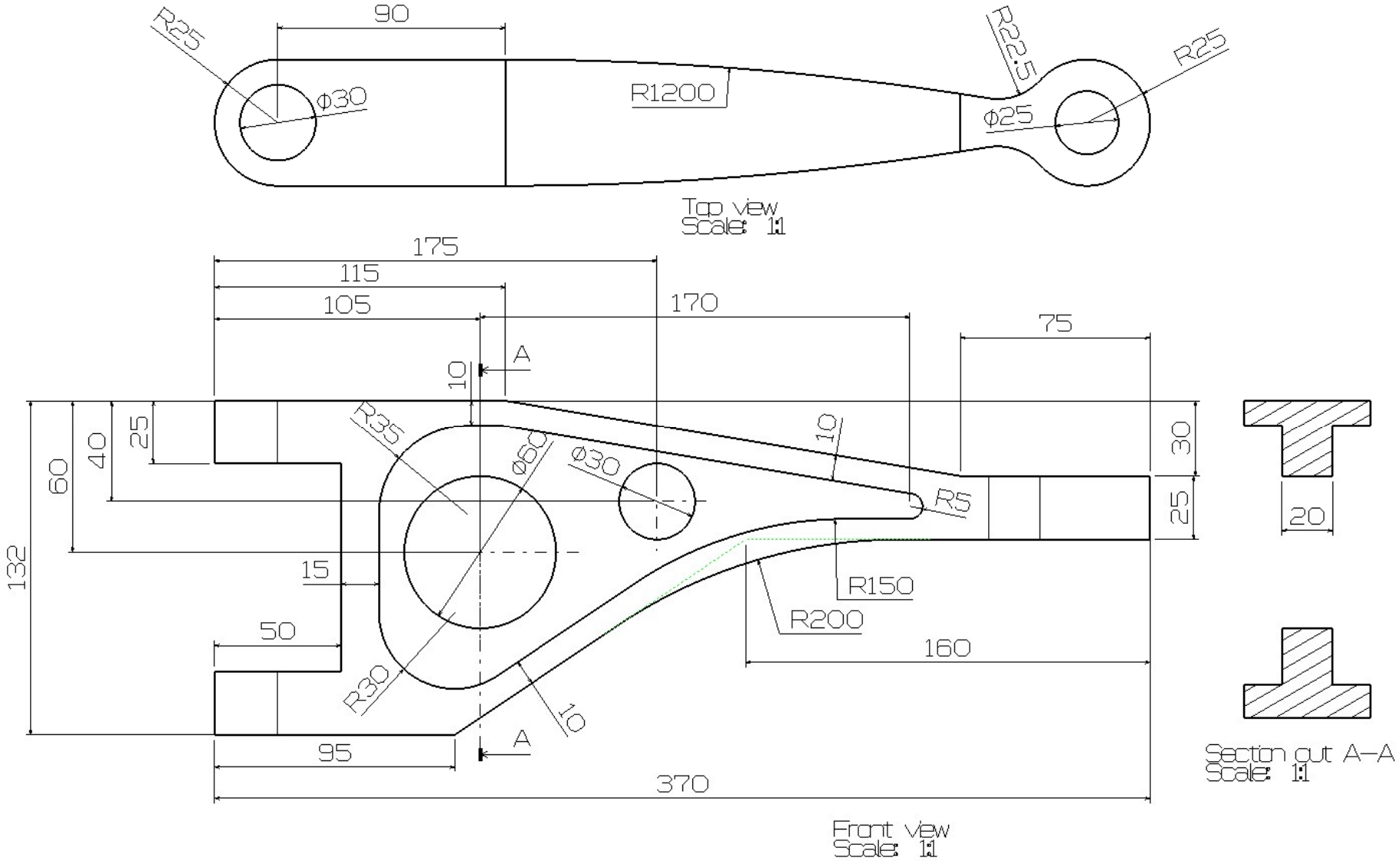
실습 과제

Cantilever



실습 과제

Cantilever

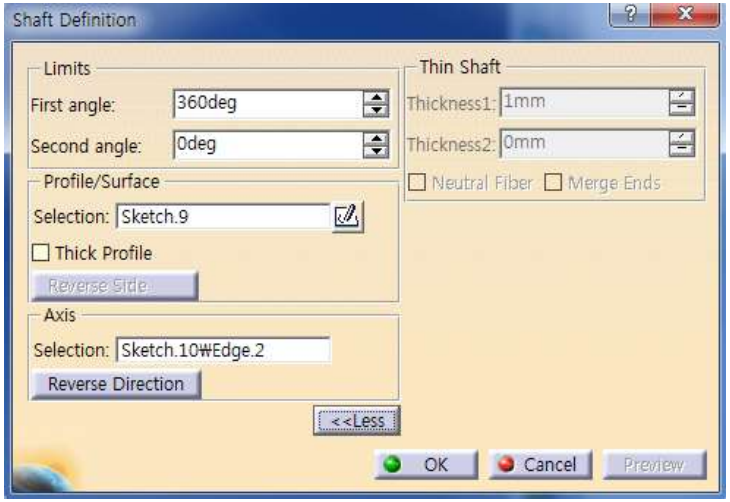


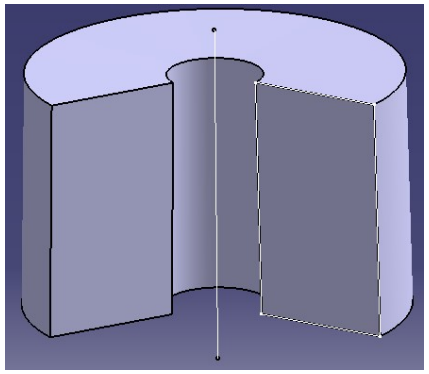
PART DESIGN TOOLS

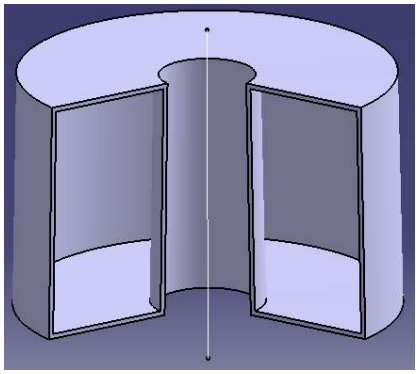
Sketch-Based Features

Shaft 

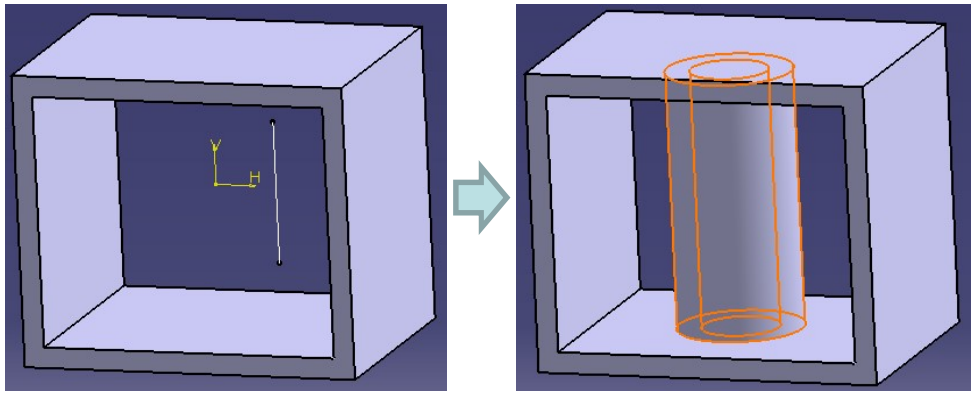
- Profile을 회전하여 모델 생성





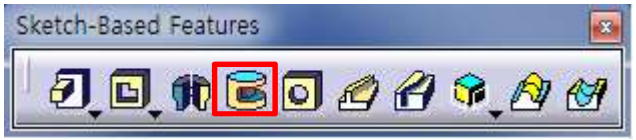


<Merge Ends(Thick)>



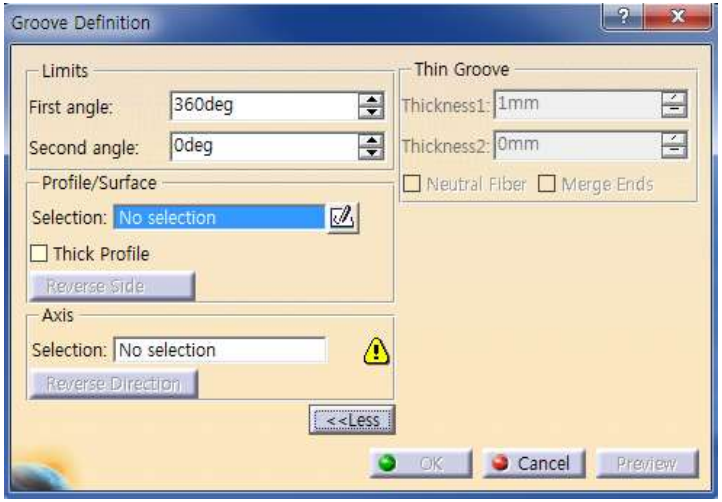
PART DESIGN TOOLS

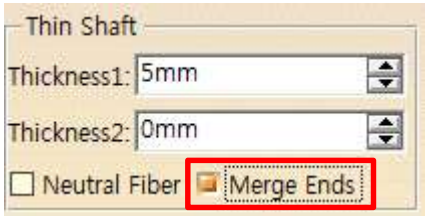
Sketch-Based Features

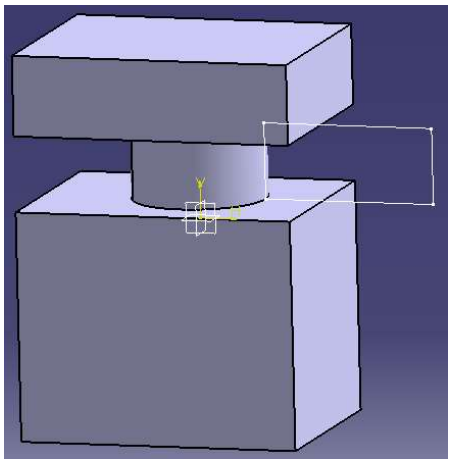


Groove

- Profile을 회전하여 모델 일부 제거







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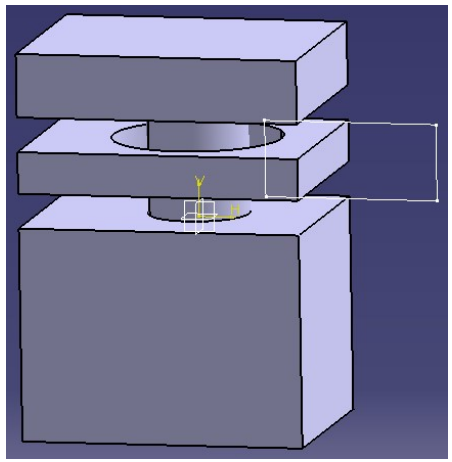
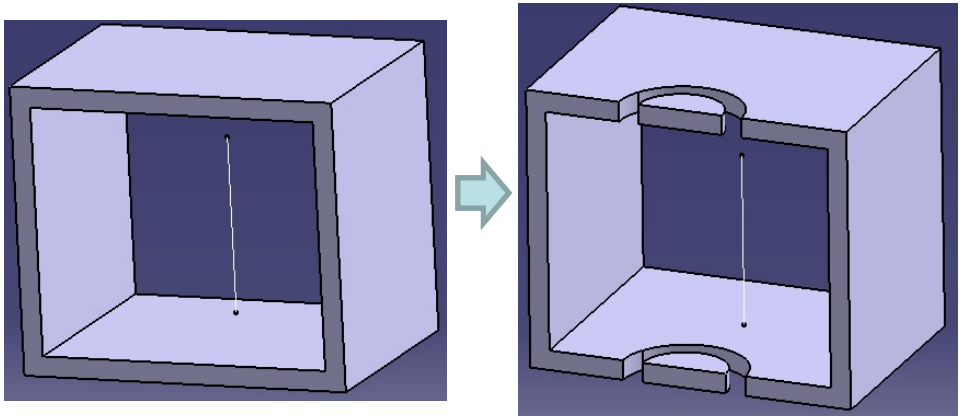


<Thick profile>

<Merge Ends(Thick)>

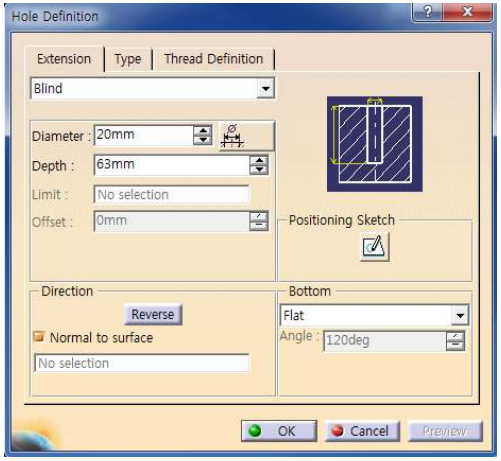


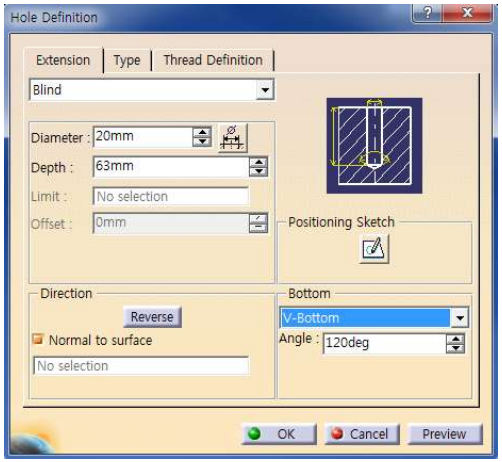
PART DESIGN TOOLS

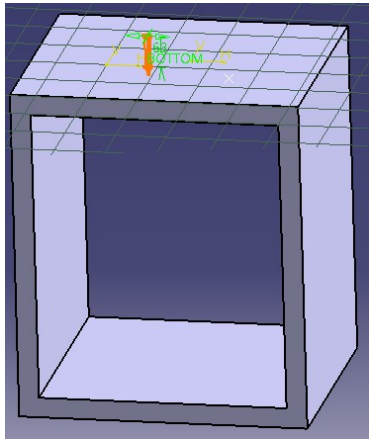
Sketch-Based Features

Hole 

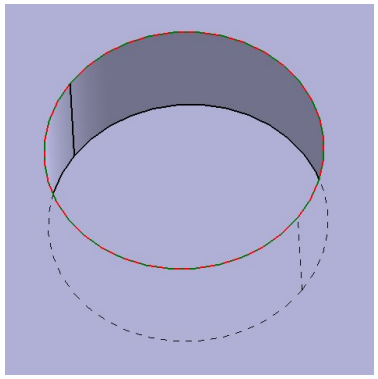
- 선택된 면에 구멍을 생성



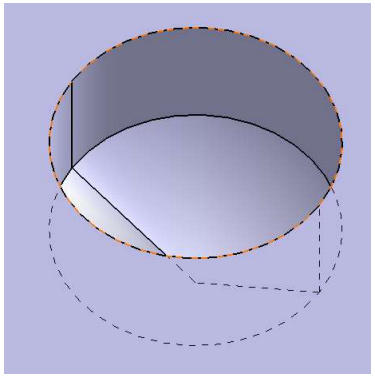




점(point)를 미리 생성하고 점을 클릭한 후 hole 기능을 사용하면 원하는 위치에 생성할 수 있다.



<Flat>



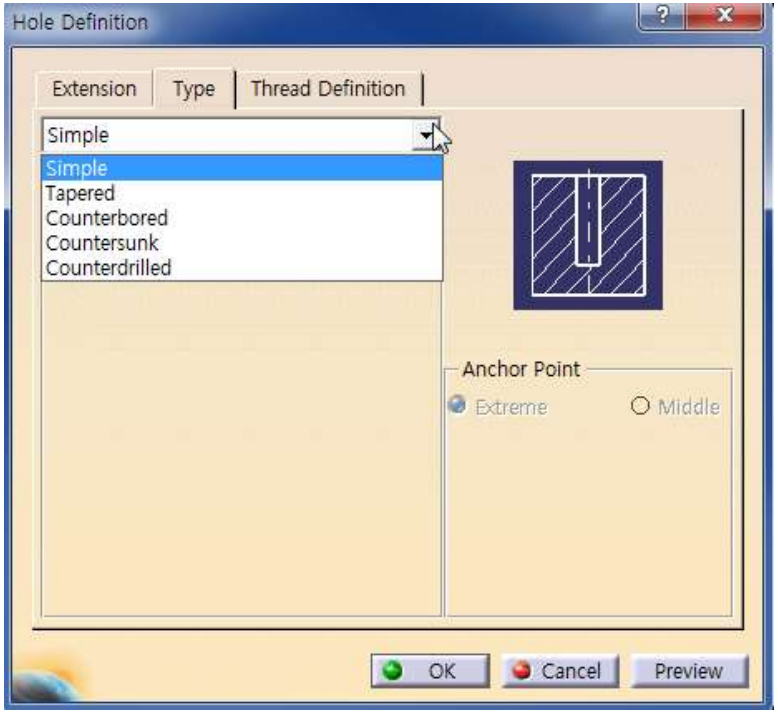
<V-Bottom>

PART DESIGN TOOLS

Sketch-Based Features

Hole

- Type : 여러 종류의 형상 선택 가능




<Sample>


<Tapered>

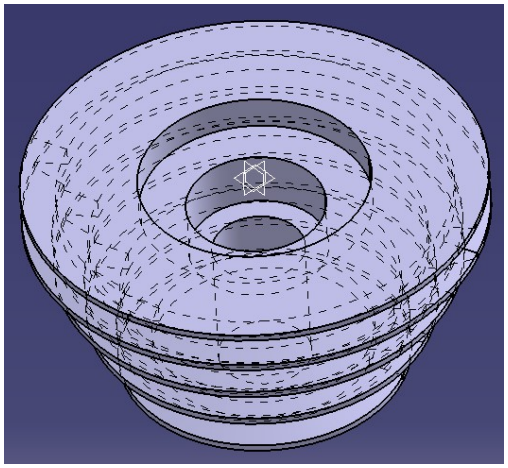

<Counterbored>

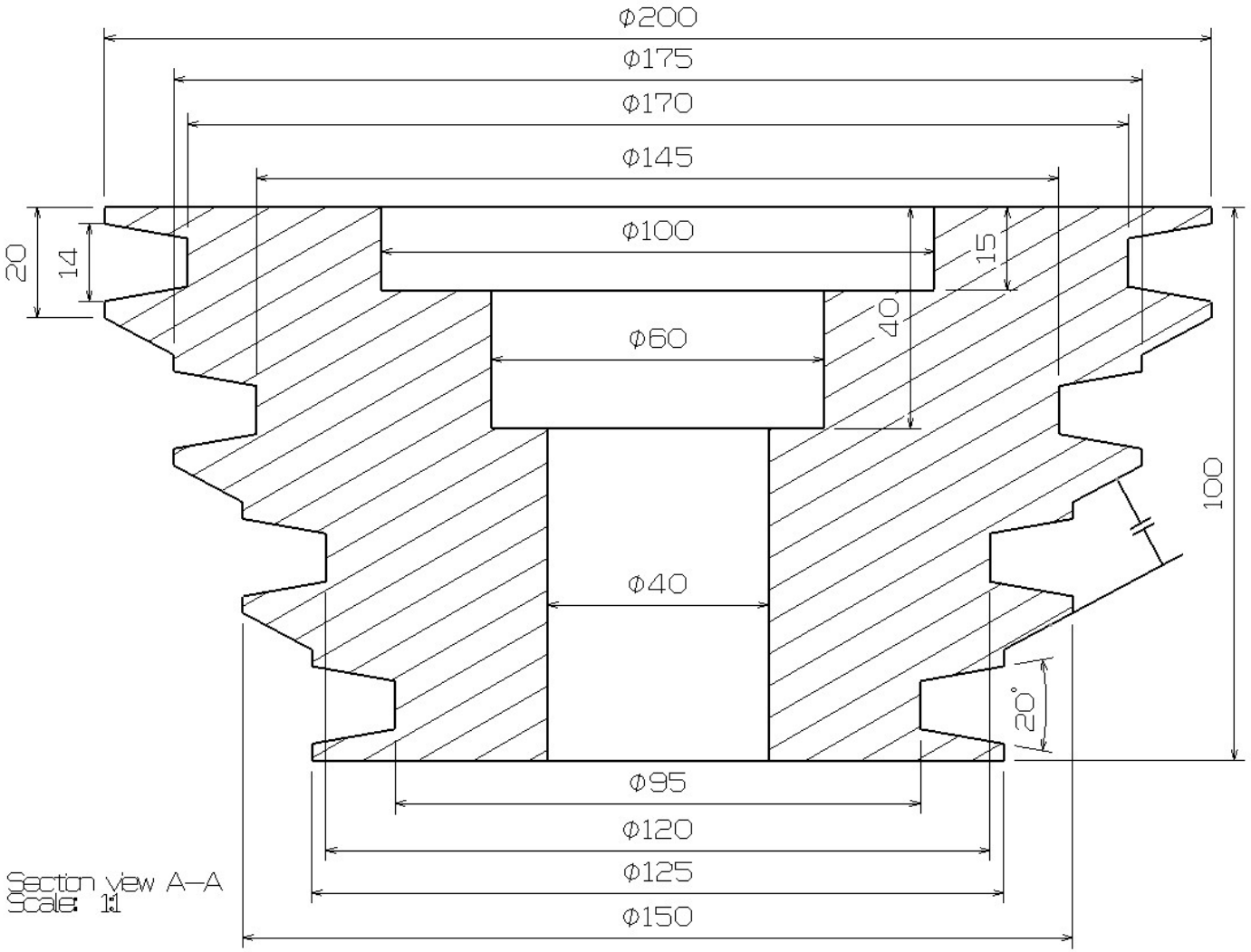

<Countersunk>


<Counterdrilled>

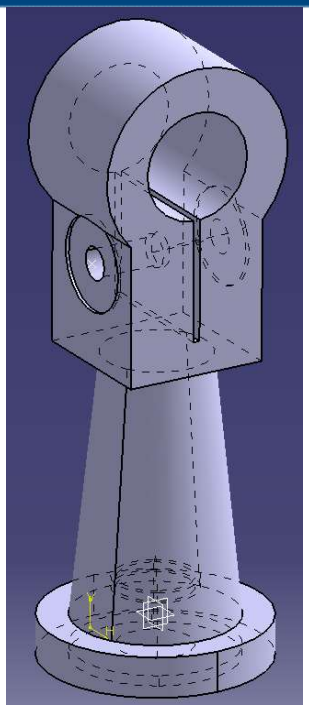
실습 예제

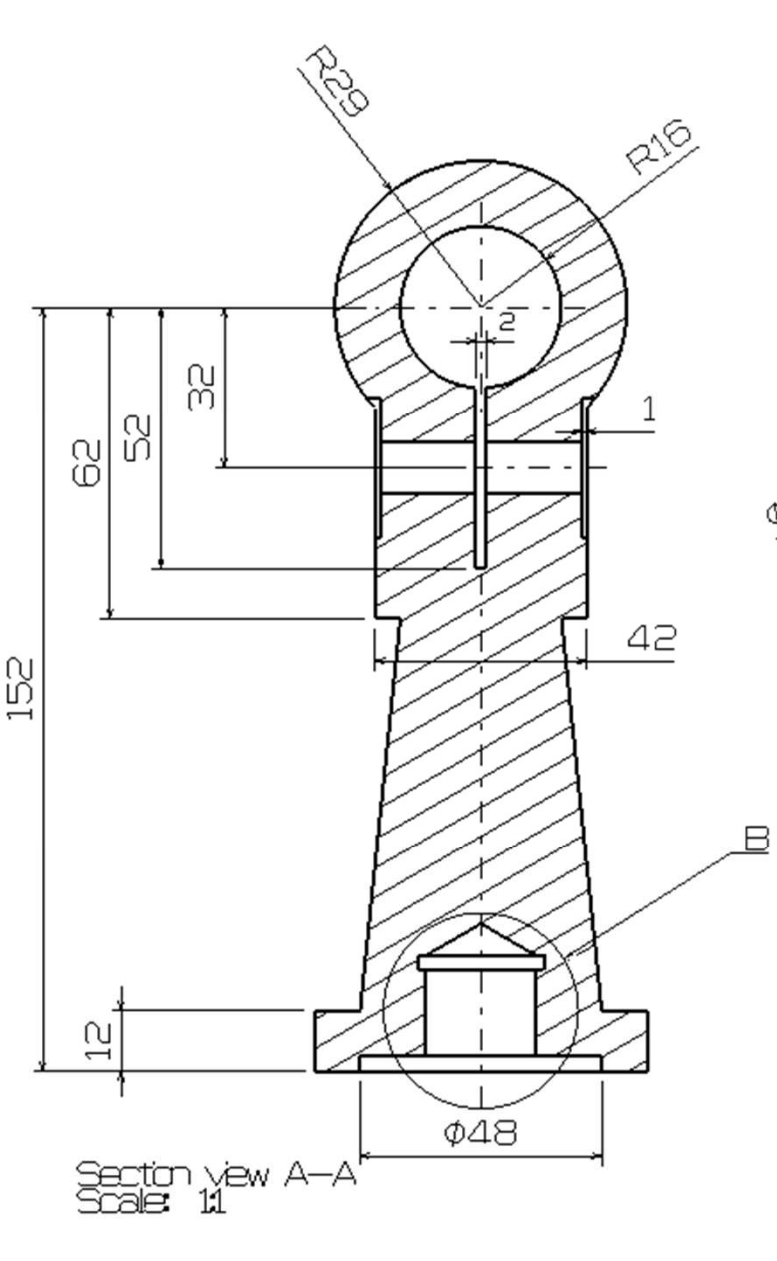
도면을 보고 3D모델을 제작하시오

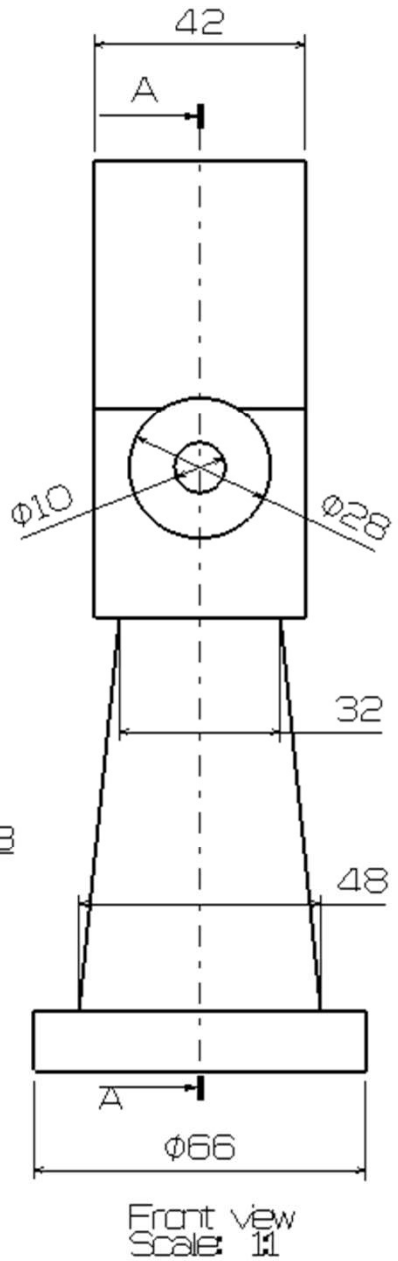


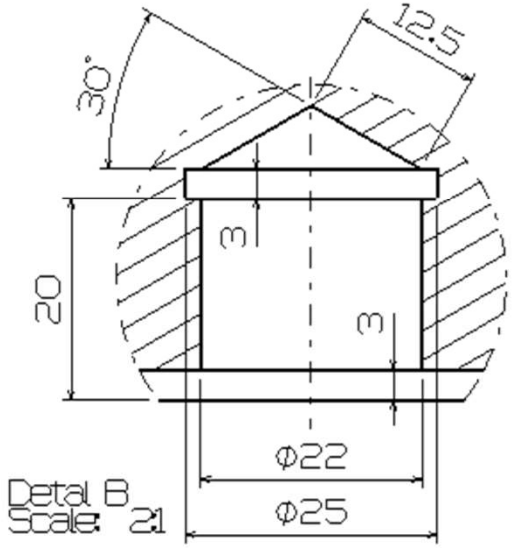


실습 예제



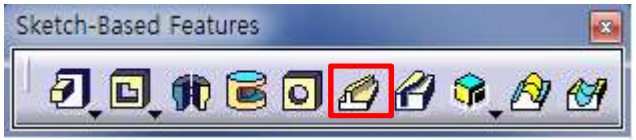






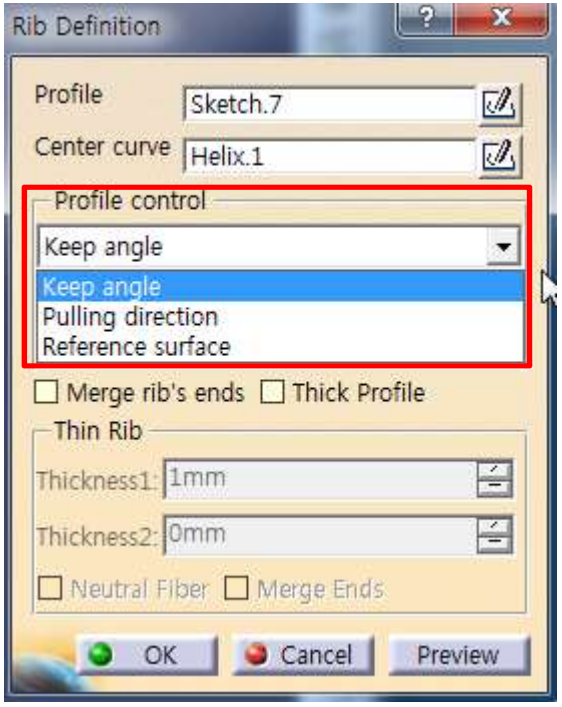
PART DESIGN TOOLS

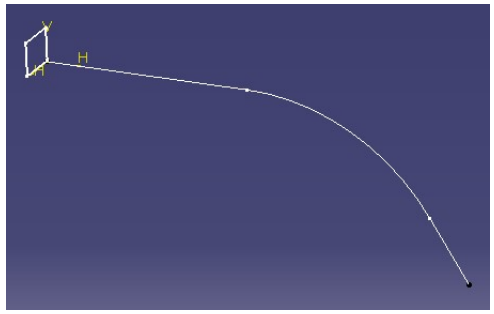
Sketch-Based Features



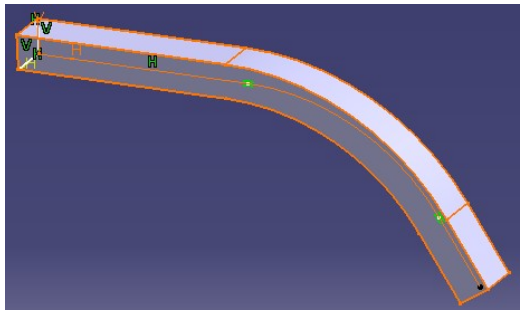
Rib

- Profile과 curve를 따라 모델 생성





<Sketch>



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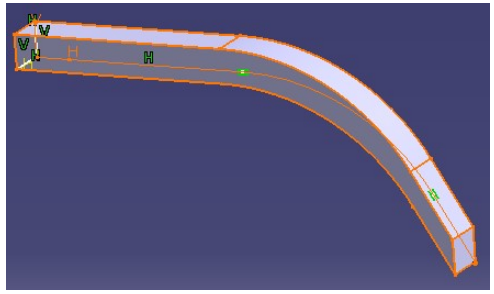
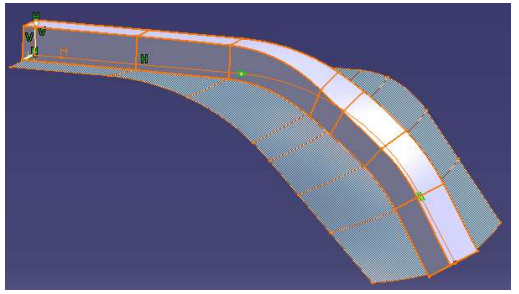


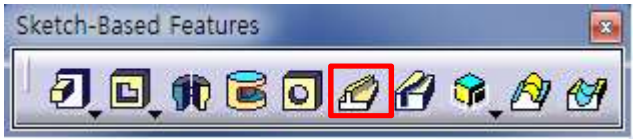
<Pulling Direction>



<Reference Surface>

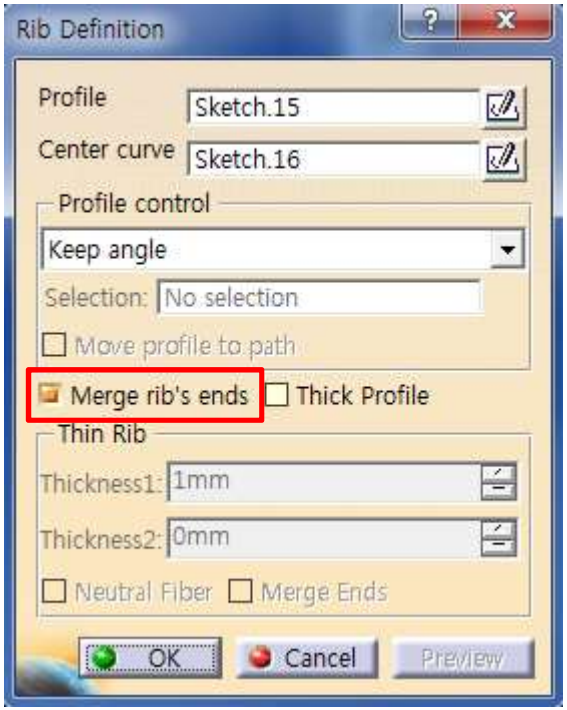
PART DESIGN TOOLS

Sketch-Based Features



Rib

- Merge rib's ends



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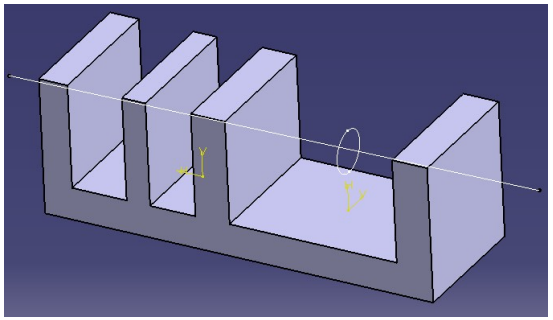
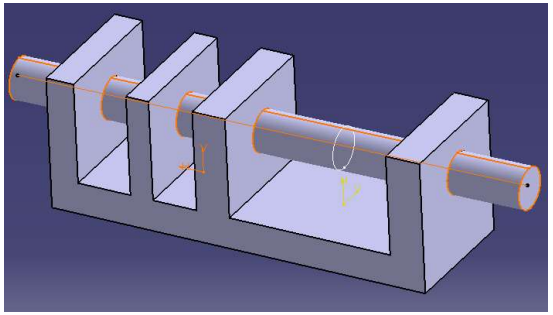
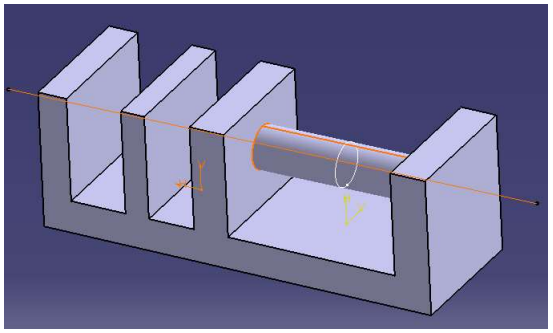


<Rib>

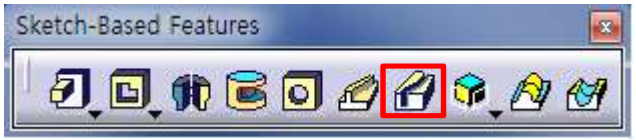


<Merge rib's ends>



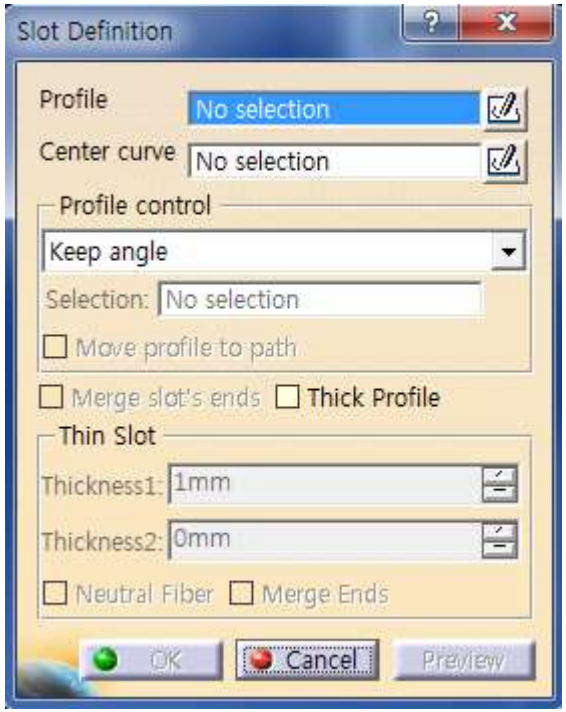
PART DESIGN TOOLS

Sketch-Based Features

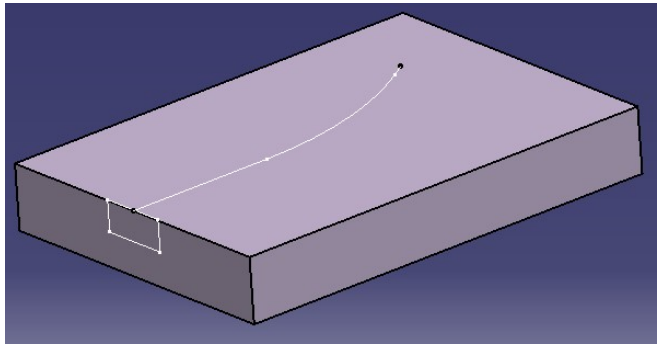


Slot

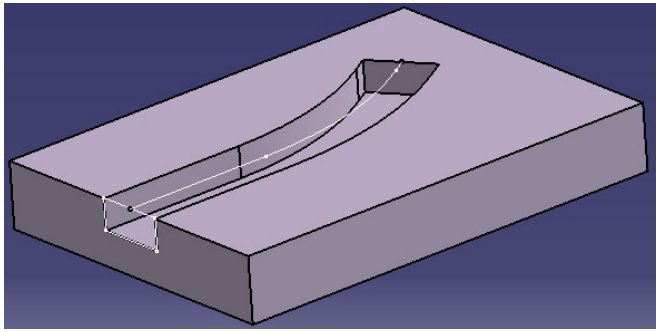
- Profile과 curve를 따라 모델 제거



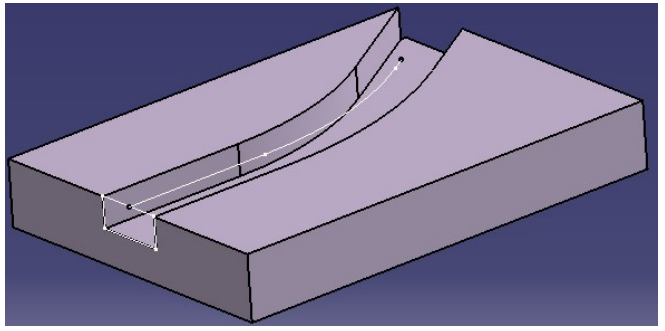
<Sketch>



<Rib>

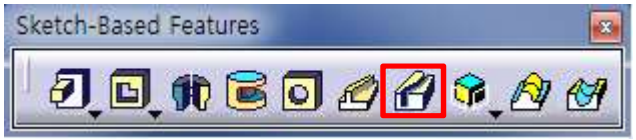


<Merge rib's ends>



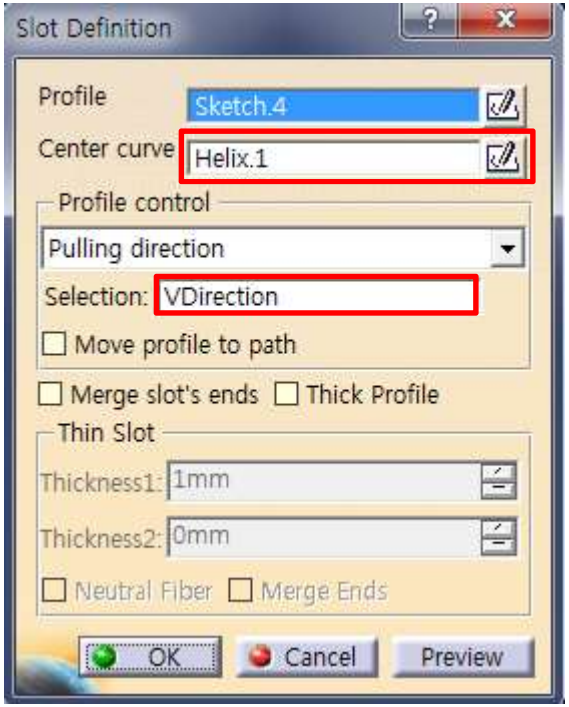
PART DESIGN TOOLS

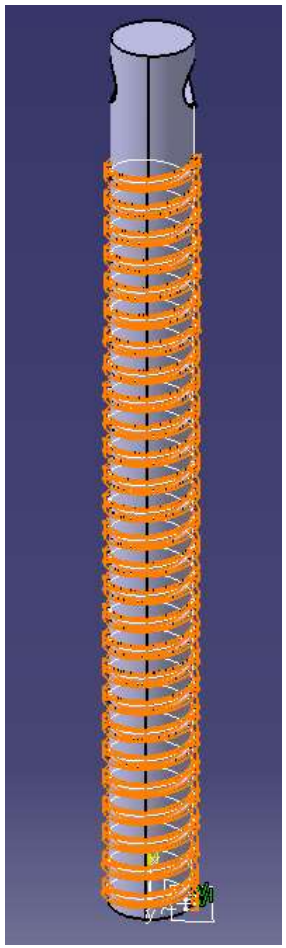
Sketch-Based Features

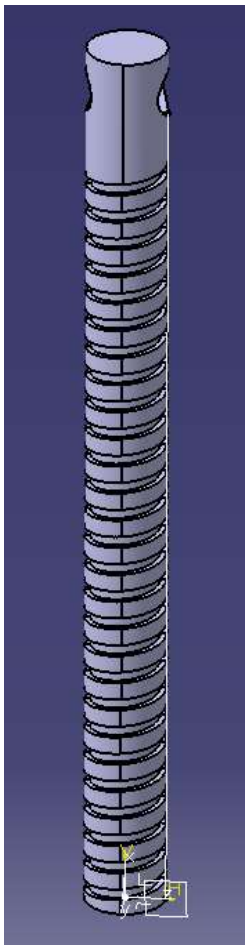


Slot

- Profile과 curve를 따라 모델 제거

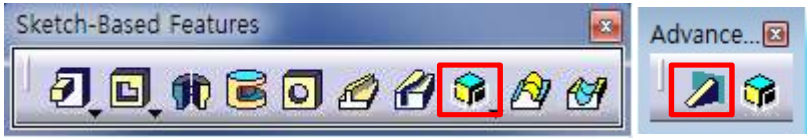






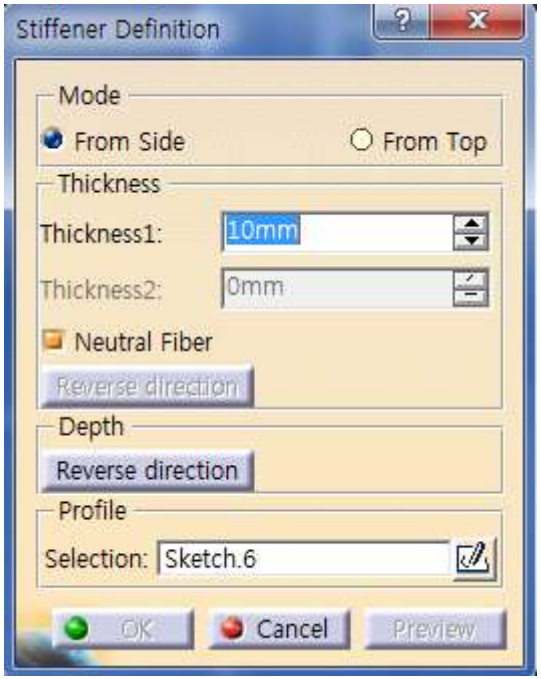
PART DESIGN TOOLS

Sketch-Based Features

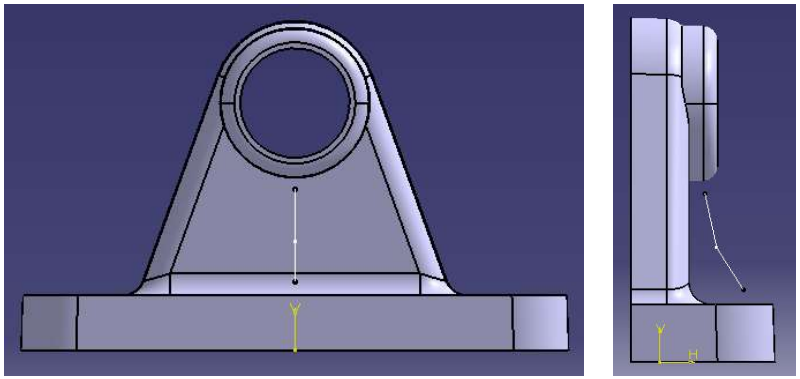


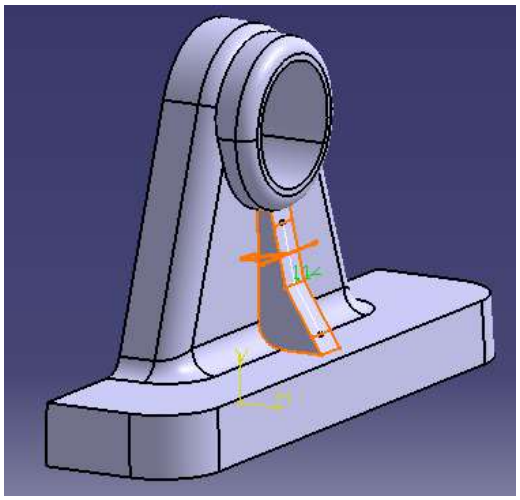
Stiffener

- 보강재 생성



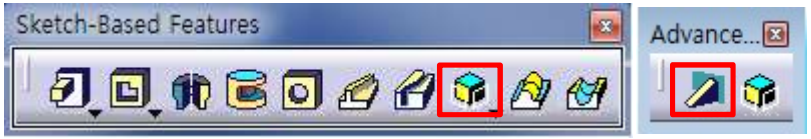
From side: 스케치에 수직한 방향으로 가상의 면을 생성하고 모델이 존재하는 방향으로 생성





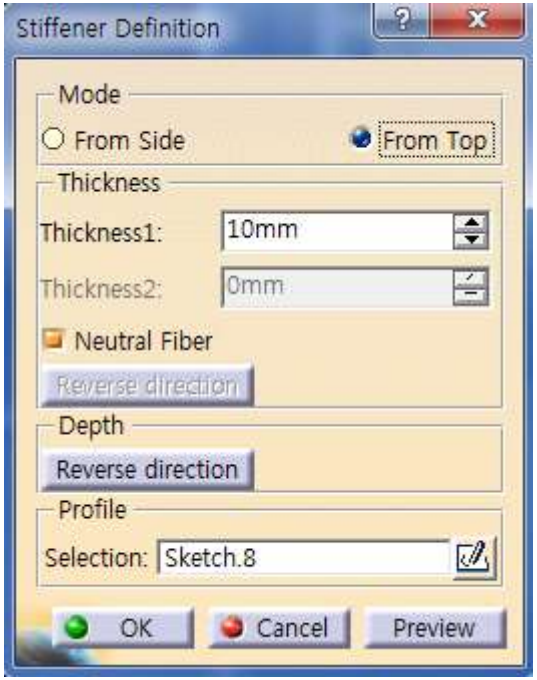
PART DESIGN TOOLS

Sketch-Based Features

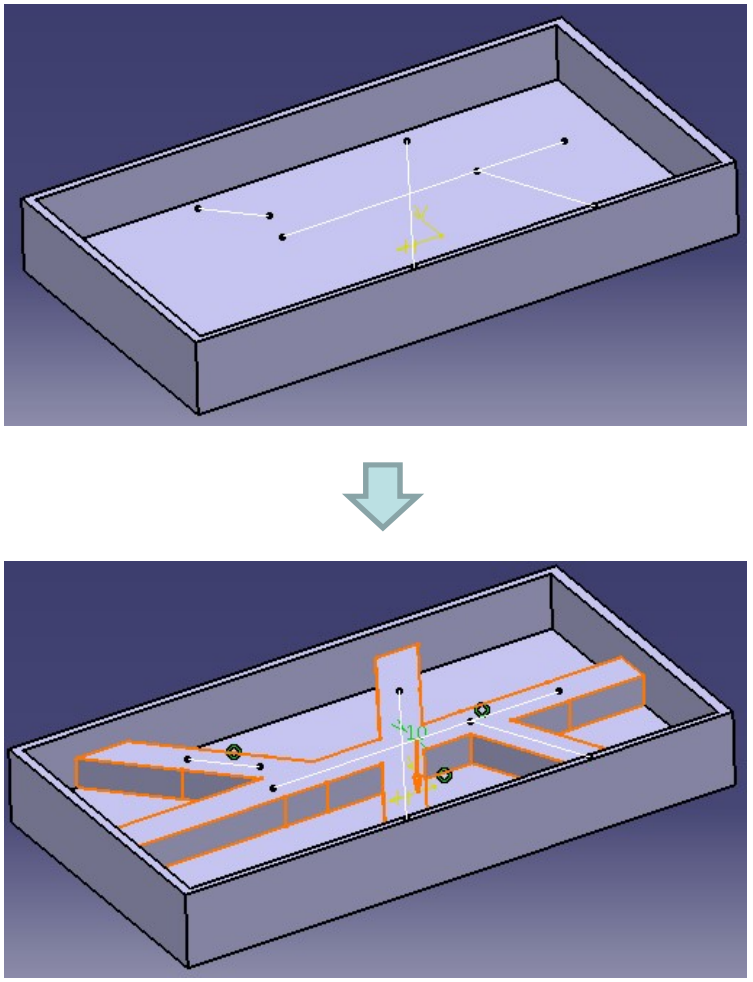


Stiffener

- 보강재 생성



From top: 스케치에 수직한 방향으로 생성



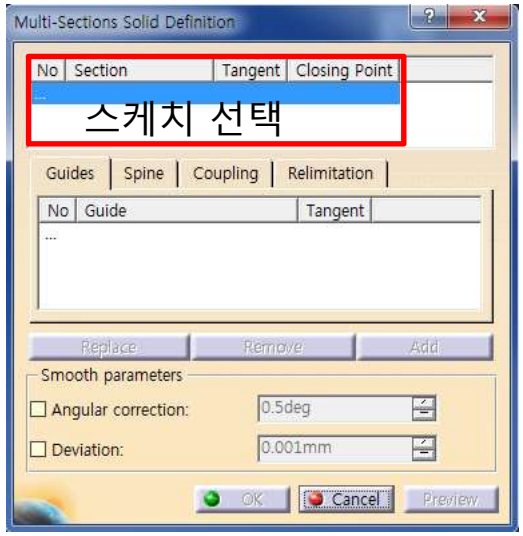
<Pad의 Merge ends를 사용해도 동일한 형상을 얻음>

PART DESIGN TOOLS

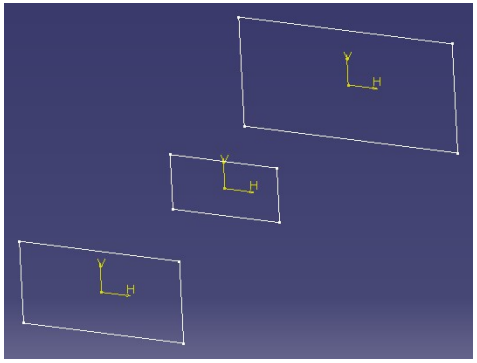
Sketch-Based Features

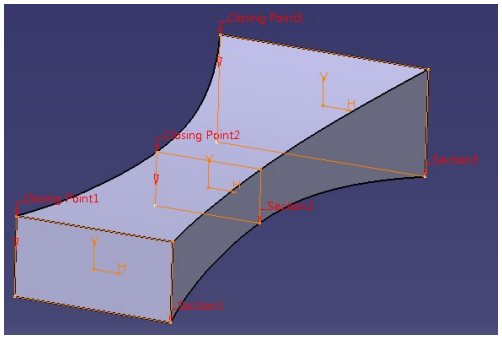
Multi-sections Solid

- 여러 면을 지나는 모델 생성

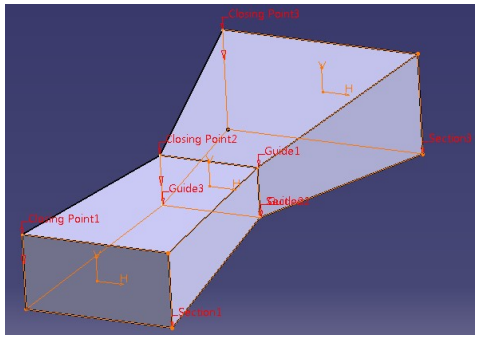


- 선택한 스케치를 순서대로 연결하여 모델 생성
- Guide를 선택할 경우 guide를 따라 생성





<Without guides>



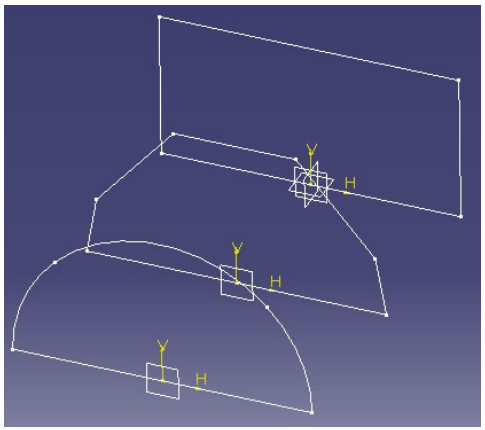
<With guides>

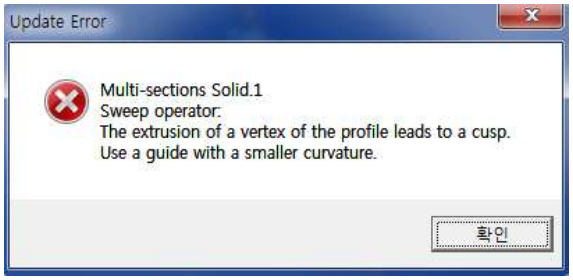
PART DESIGN TOOLS

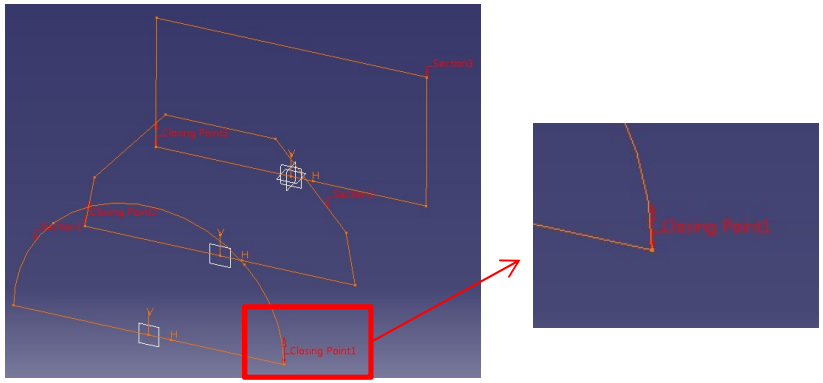
Sketch-Based Features

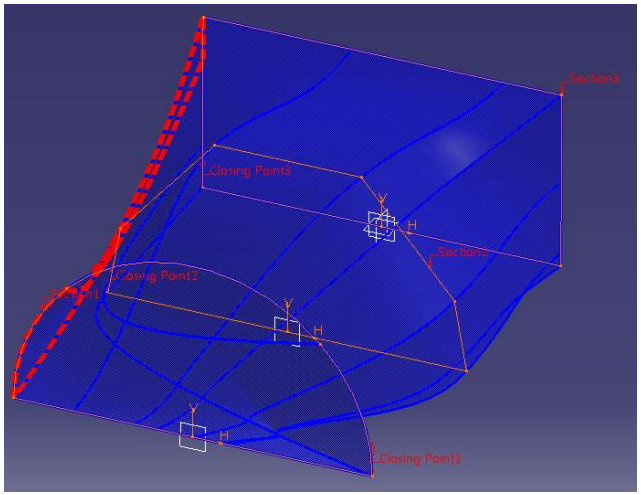
Multi-sections Solid

- Closing Point 가 동일선상에 없을 경우 형상이 꼬임









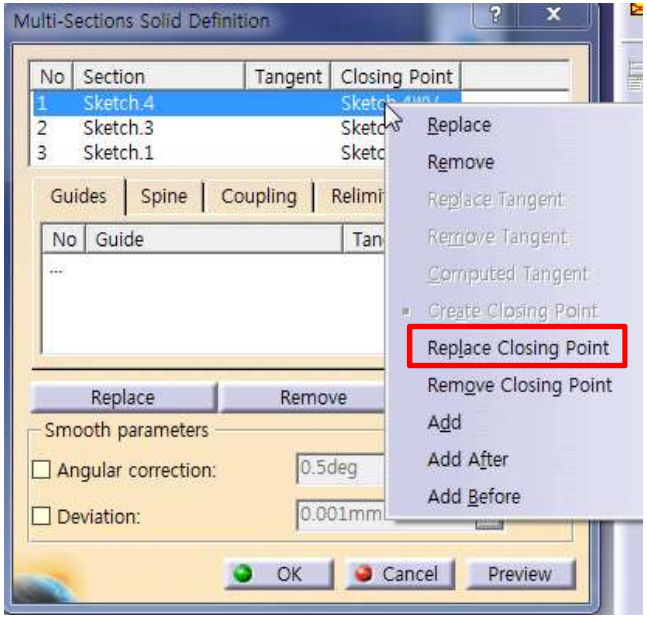
PART DESIGN TOOLS

Sketch-Based Features

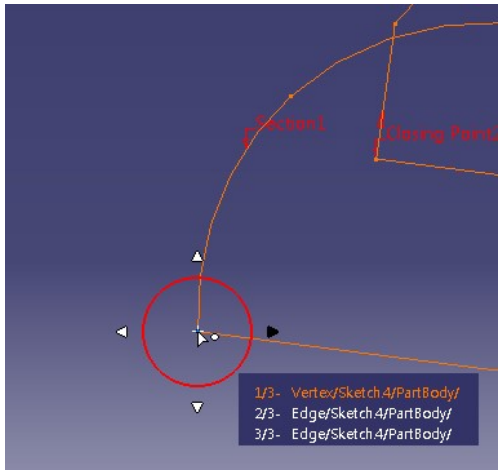
Multi-sections Solid 

- Closing Point 수정 방법

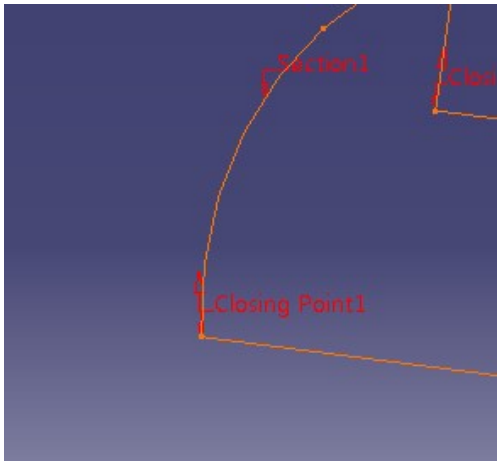
수정이 필요한 Profile에서 오른쪽 클릭 후 Replace Closing Point 클릭



동일선상에 있는 점에 마우스를 대고 키보드 ↑(또는↓) 버튼을 이용



Closing Point 생성 확인 후 방향까지 확인

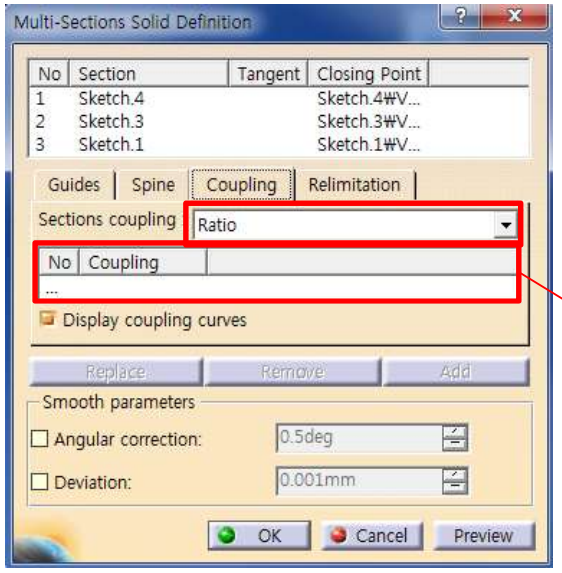


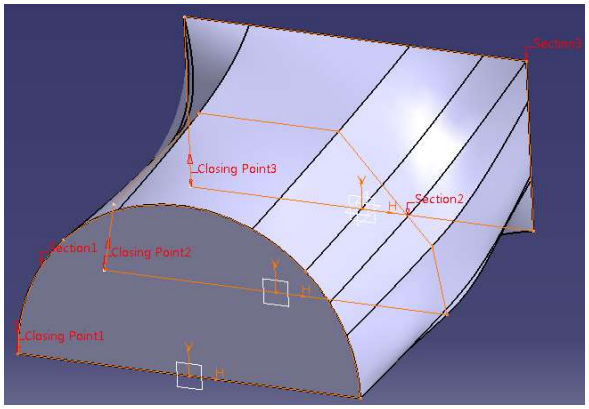
PART DESIGN TOOLS

Sketch-Based Features

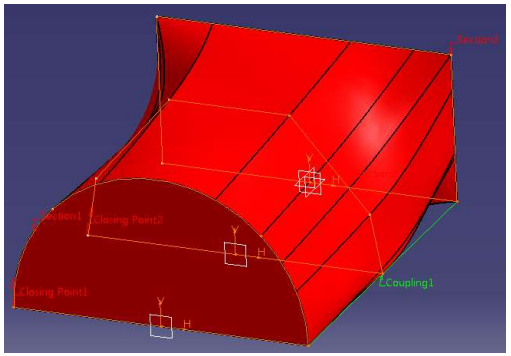
Multi-sections Solid

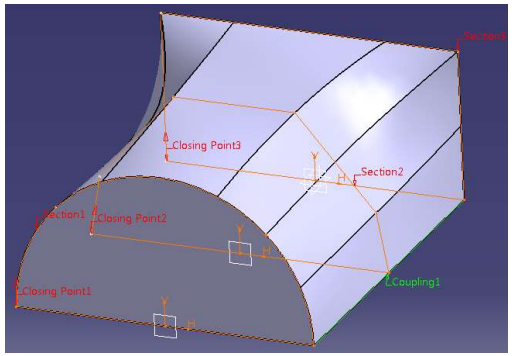
- Closing Point 수정 후 Coupling의 Ratio 선택
  (다른 타입은 형상의 꼭지점 수가 같아야 함)





Coupling point를 선택하여 좀 더 형상 수정 가능



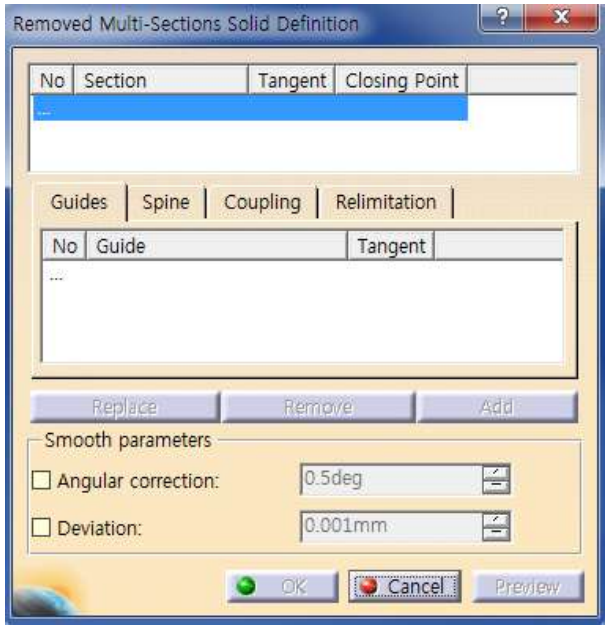


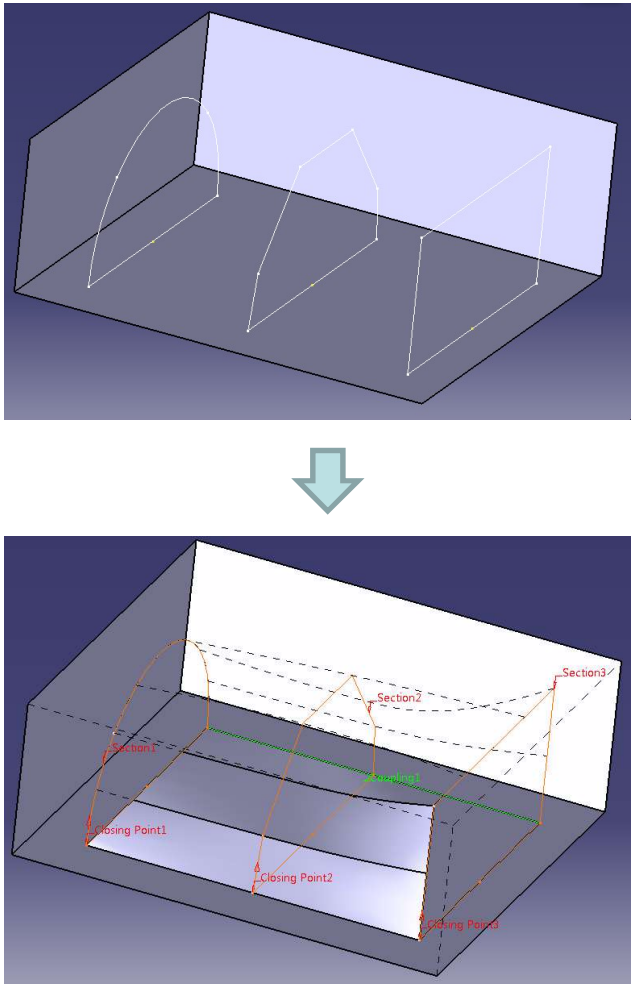
PART DESIGN TOOLS

Sketch-Based Features

Remove Multi-sections Solid

- 여러 면으로 생성된 모델의 부피만큼 제거
- Multi-sections Solid와 동일한 방법으로 진행

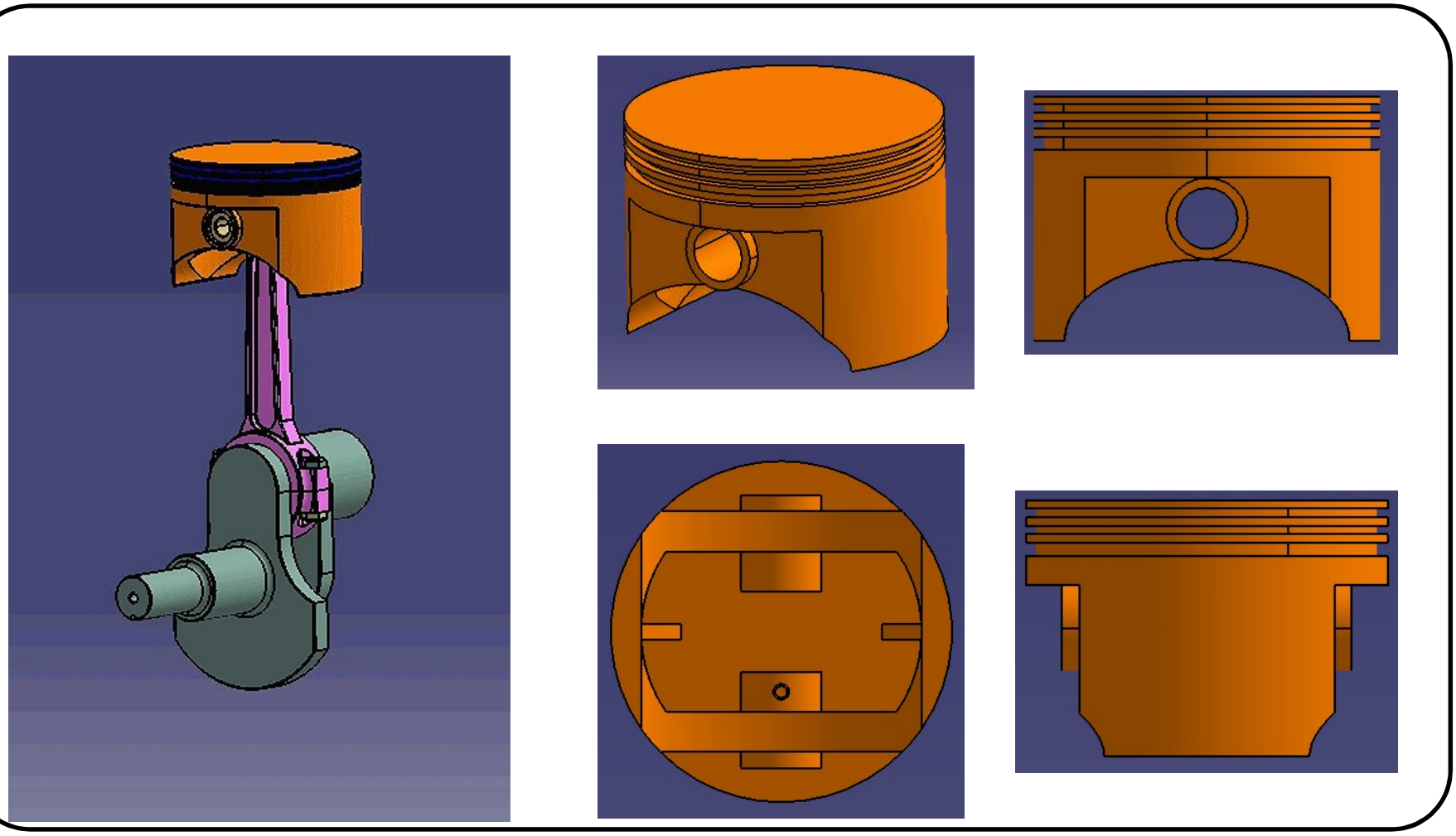




실습 과제

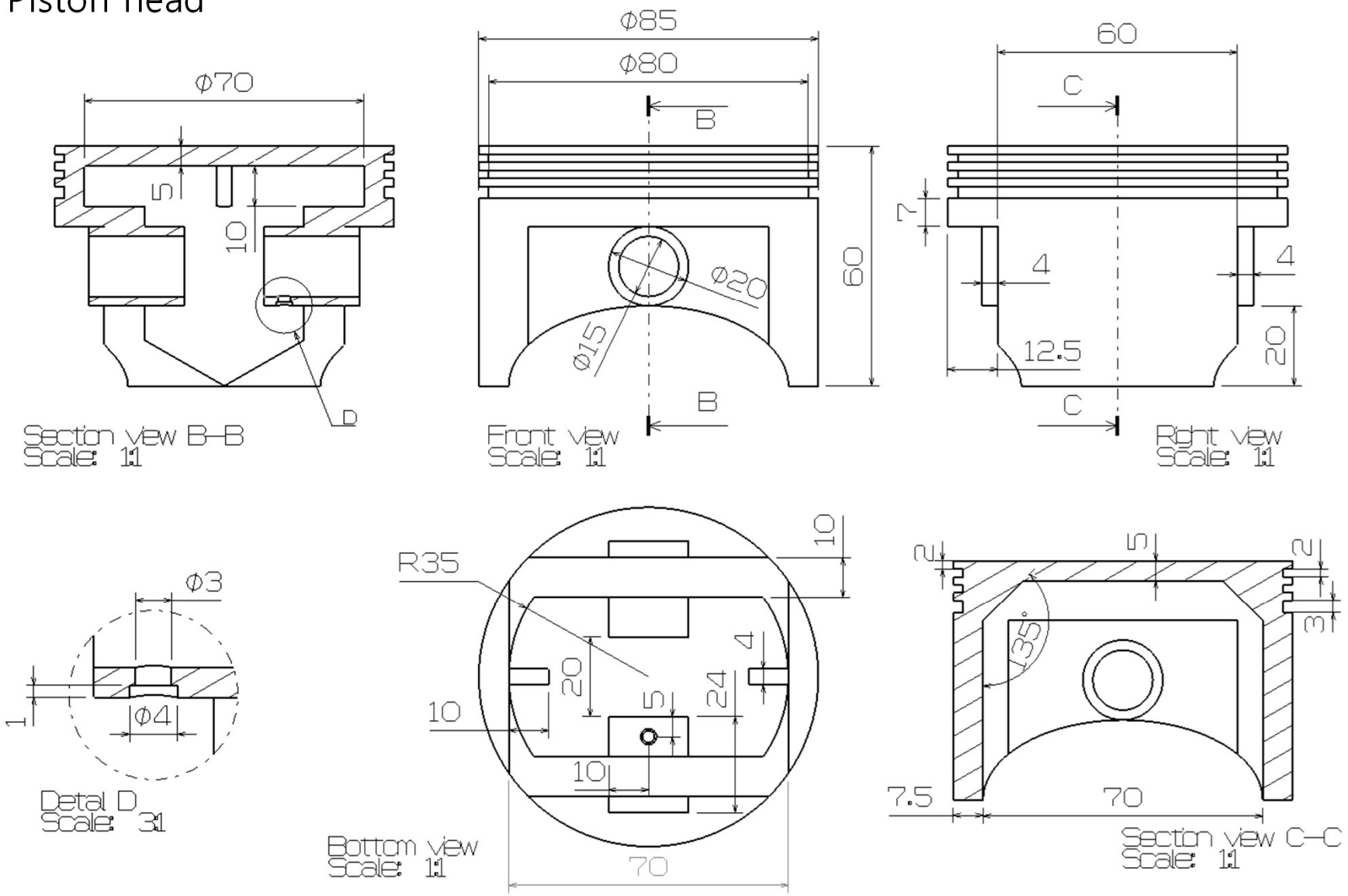
Piston Head

- 피스톤의 가스 압력을 받아 수직 운동하는 머리부분.



실습 과제

Piston head



CONTENTS

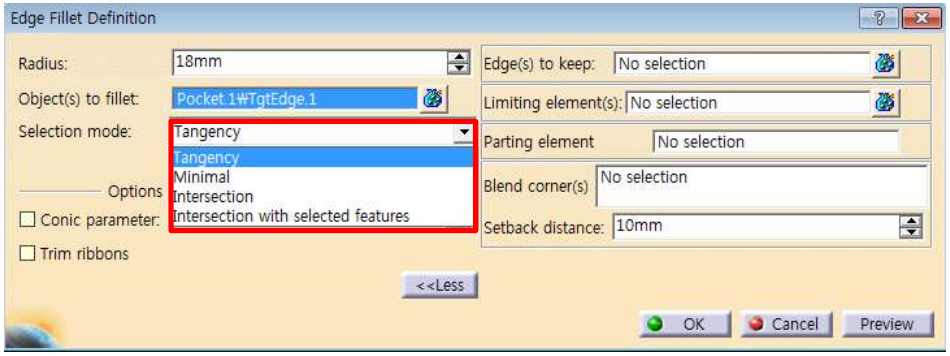
- ✓ 시작하기
- ✓ Reference Elements
- ✓ Sketch-Based Features
- ✓ **Dress-Up Features**
- ✓ Transformation Features
- ✓ Boolean Operation

PART DESIGN TOOLS

Dress-Up Features

Edge Fillet 

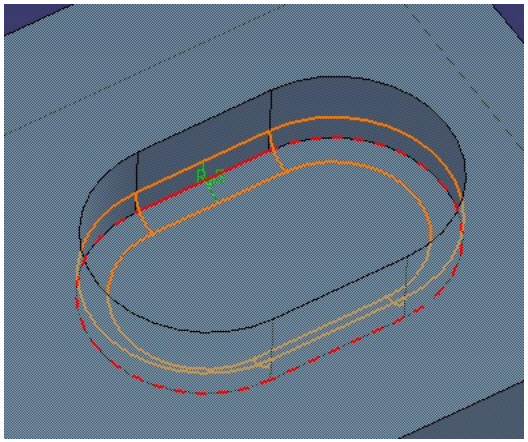
- 모서리를 원호 형태로 제거/생성함



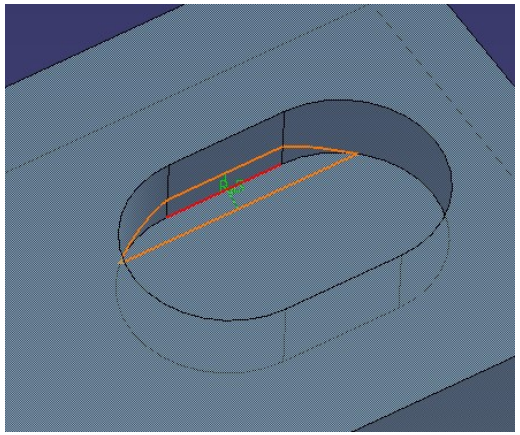
Tangency: tangency로 연결된 라인을 모두 fillet 생성

Minimal: 선택한 라인과 라인의 연장선까지 fillet 생성

<Tangency>



<Minimum>

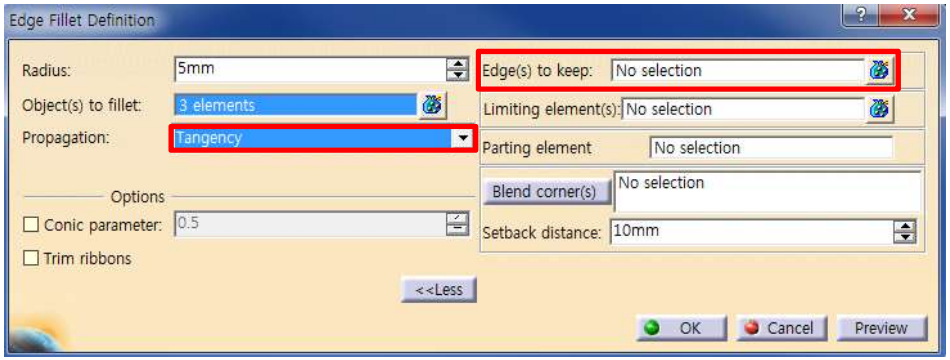


PART DESIGN TOOLS

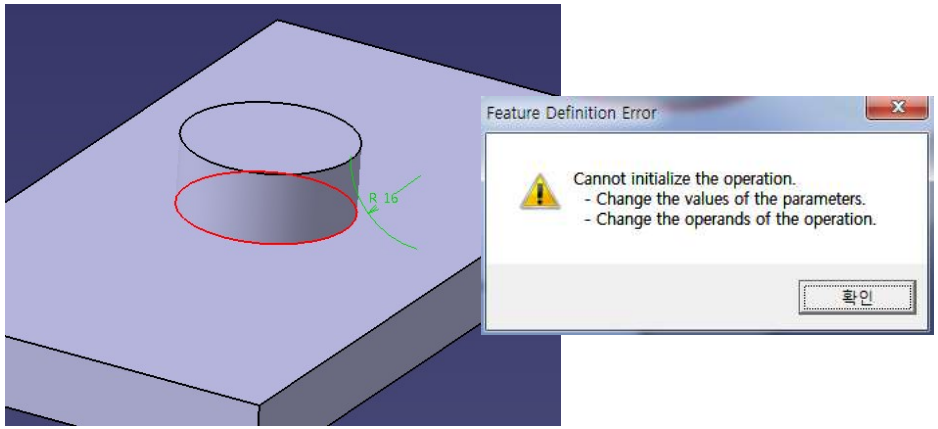
Dress-Up Features

Edge Fillet

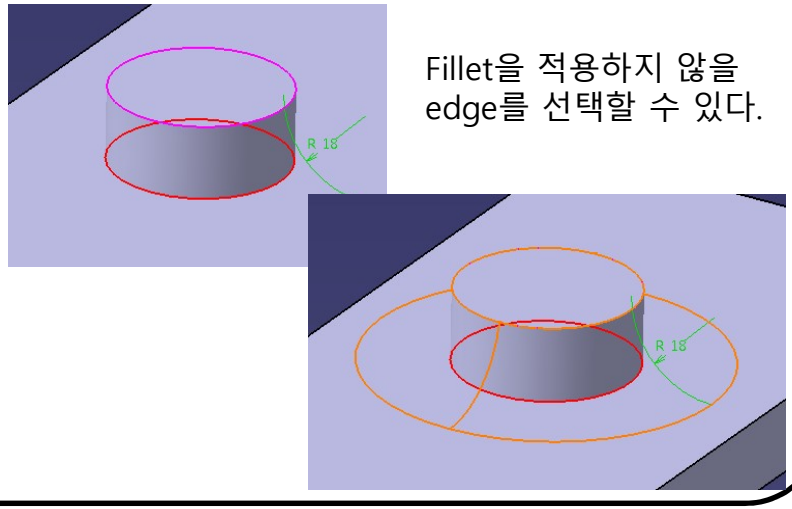
- 모서리를 원호 형태로 제거/생성함



<Without 'edge to keep'>



<With 'edge to keep'>



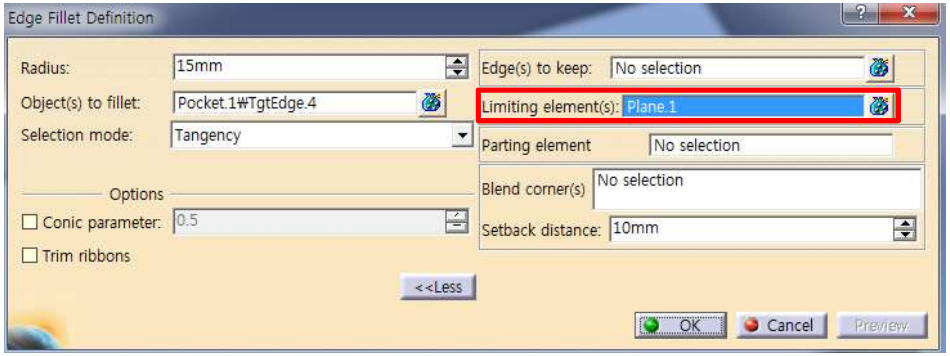
Fillet을 적용하지 않을 edge를 선택할 수 있다.

PART DESIGN TOOLS

Dress-Up Features

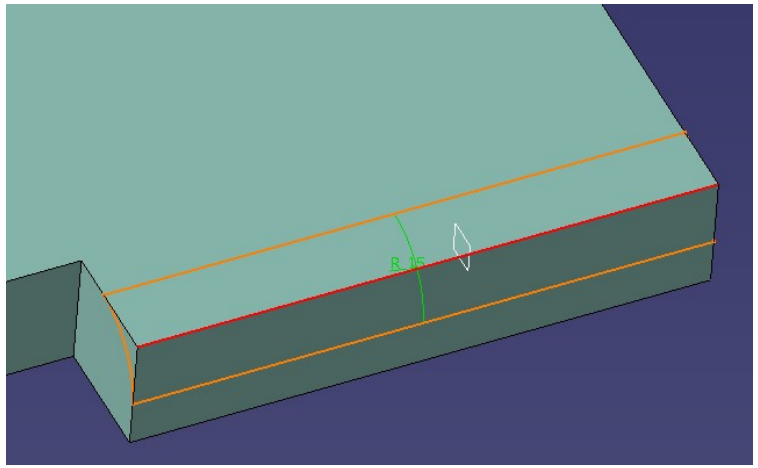
Edge Fillet

- 모서리를 원호 형태로 제거/생성함

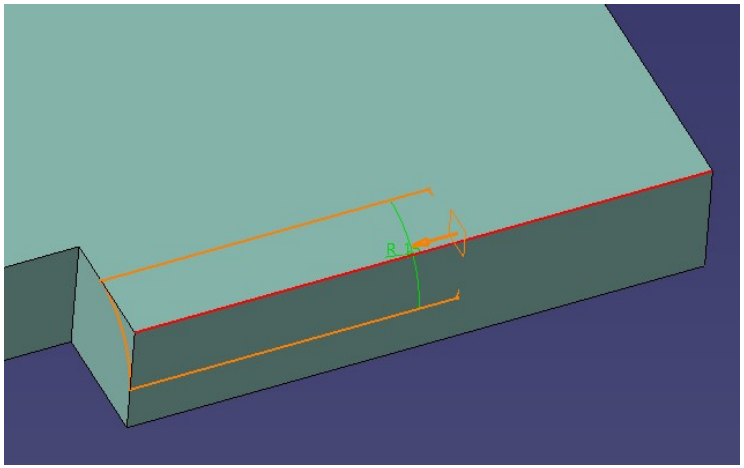


특정 면(plane)과 같은 element를 이용하여 fillet의 진행을 제한 할 수 있다.

<Without 'limiting element'>



<With 'limiting element'>

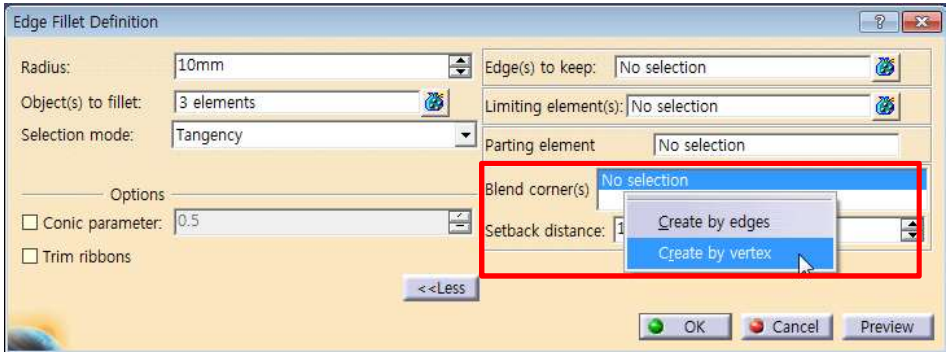


PART DESIGN TOOLS

Dress-Up Features

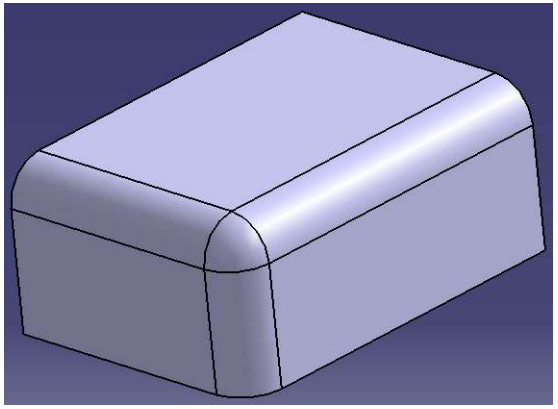
Edge Fillet

- 모서리를 원호 형태로 제거/생성함

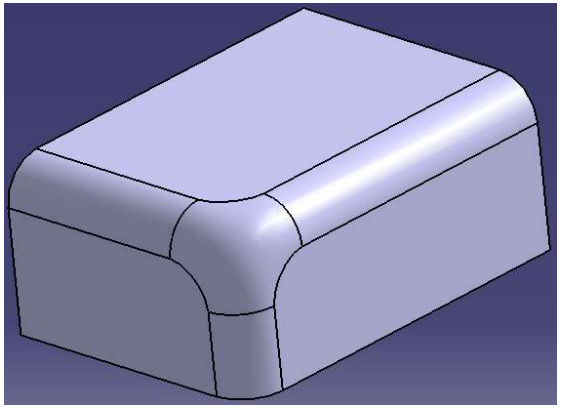


꼭지점을 기준으로 일정한 거리를 갖도록 fillet을 적용 할 수 있다.

<Edge Fillet>



<Blend corner>

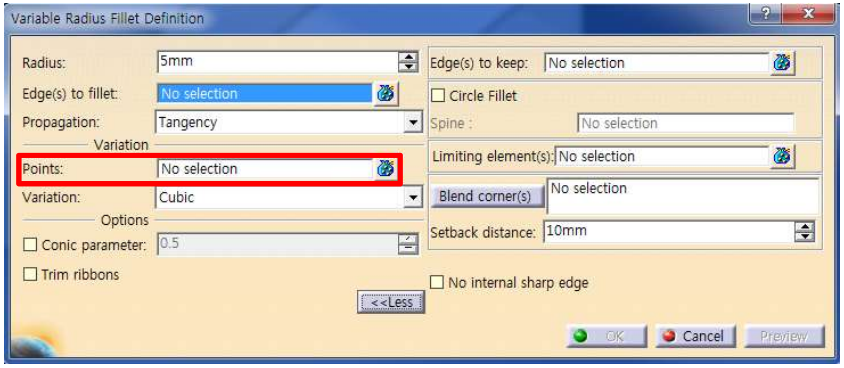


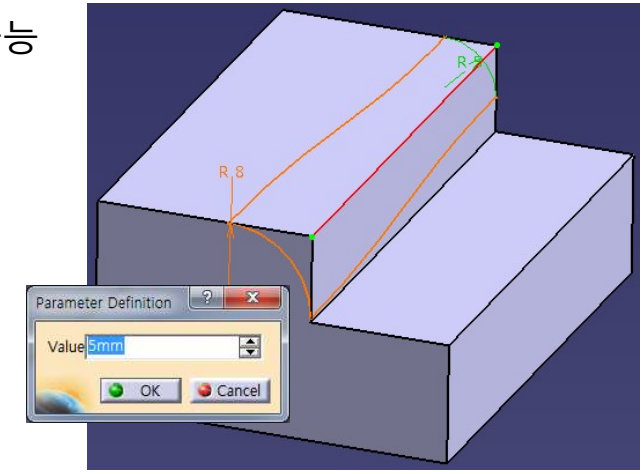
PART DESIGN TOOLS

Dress-Up Features

Variable Radius Fillet

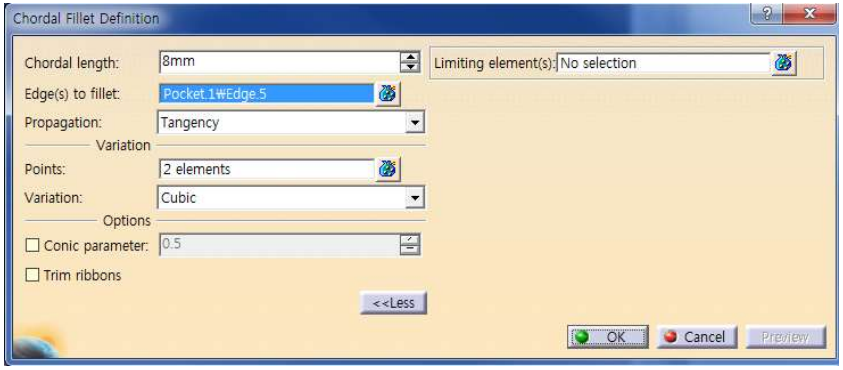
- 치수를 더블 클릭하여 여러 개의 반지름의 값을 조절 가능
- 점을 추가하여 2개 이상의 반지름 적용 가능

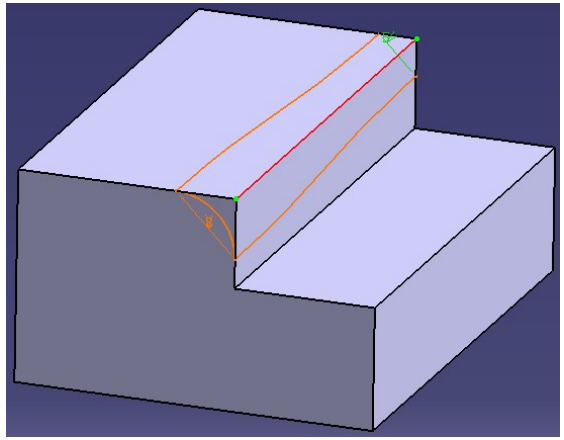




Chordal Fillet

- 반지름이 아니라 거리로 계산함(그 외에 위와 동일)



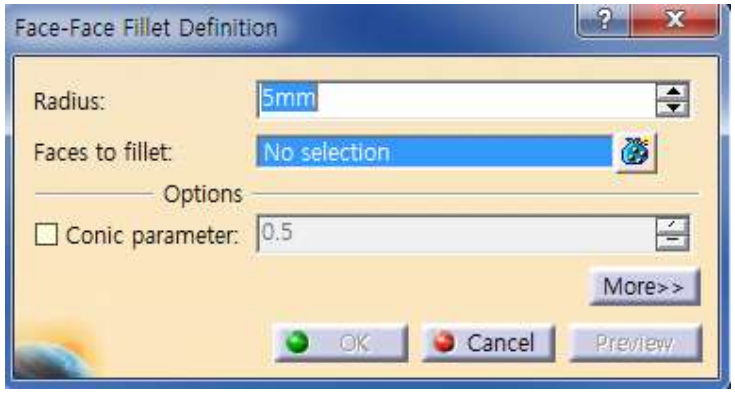


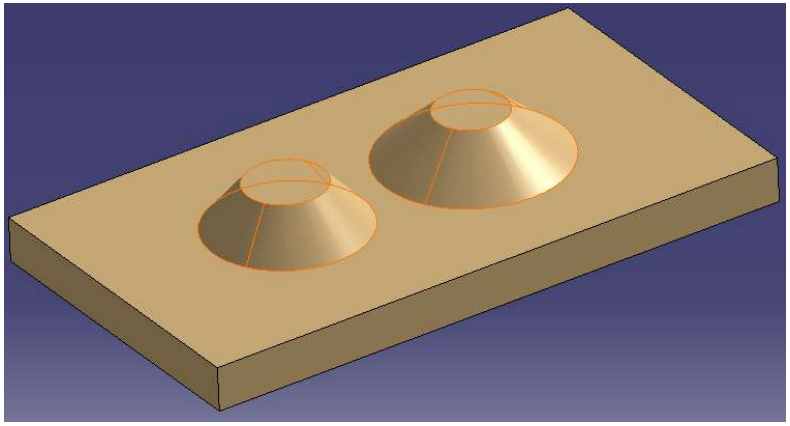
PART DESIGN TOOLS

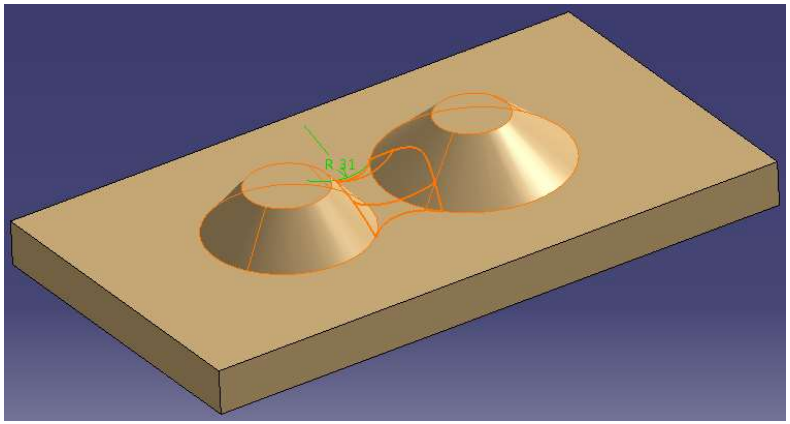
Dress-Up Features

Face-Face Fillet

- 면 사이를 fillet 연결함





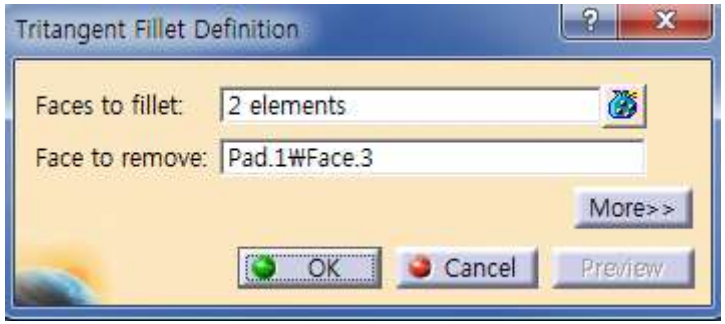


PART DESIGN TOOLS

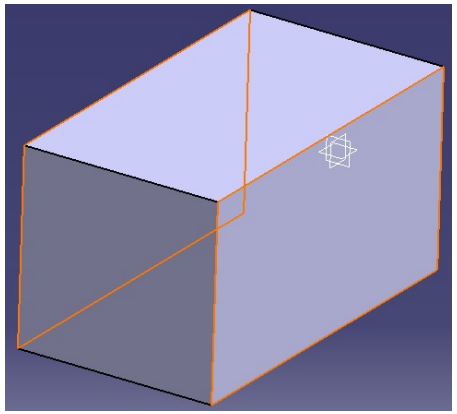
Dress-Up Features

Tritangent Fillet 

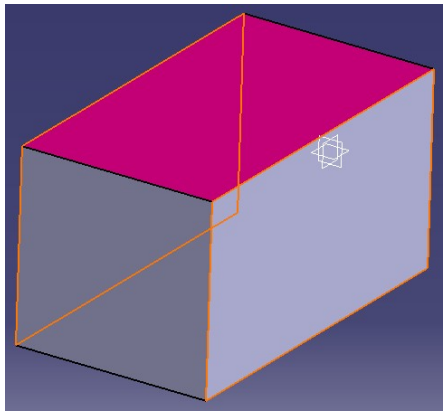
- Fillet으로 새로운 면 생성

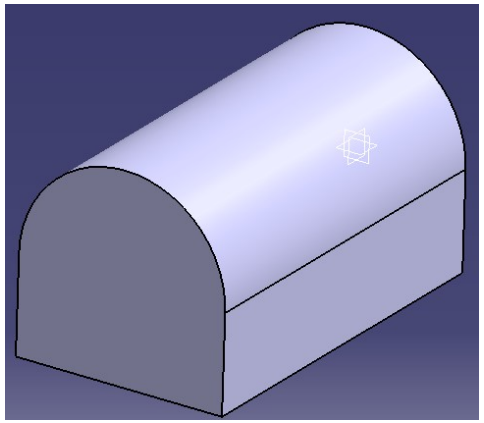


Fillet을 적용할 두 면 선택



Fillet으로 제거될 면 선택



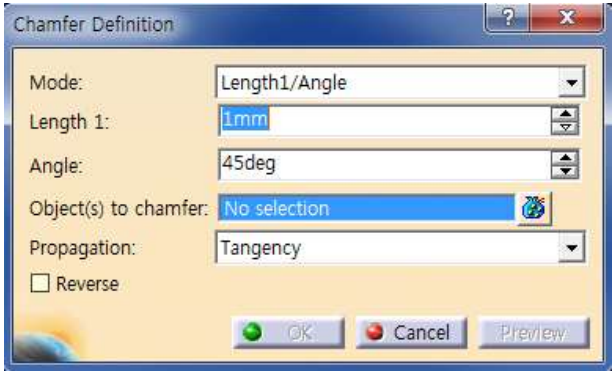


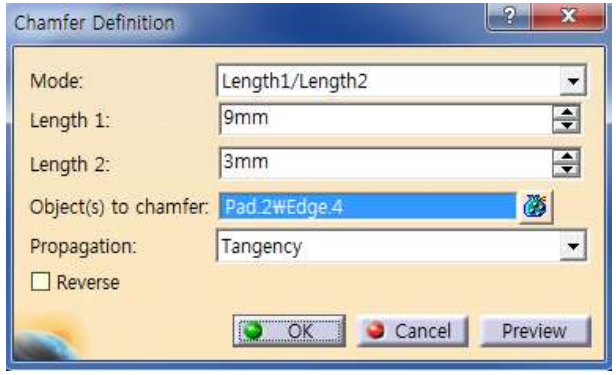
PART DESIGN TOOLS

Dress-Up Features

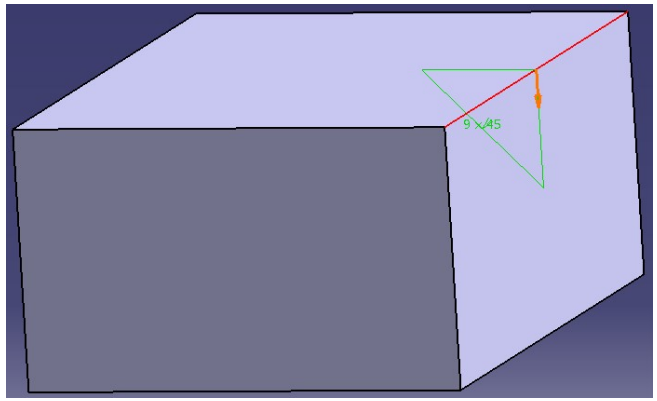
Chamfer 

- 모서리를 삼각형 형태로 수정함

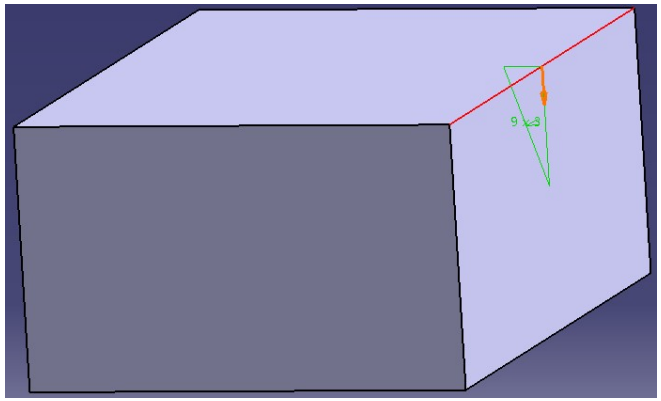




<Length1/Angle>



< Length1/Length2>

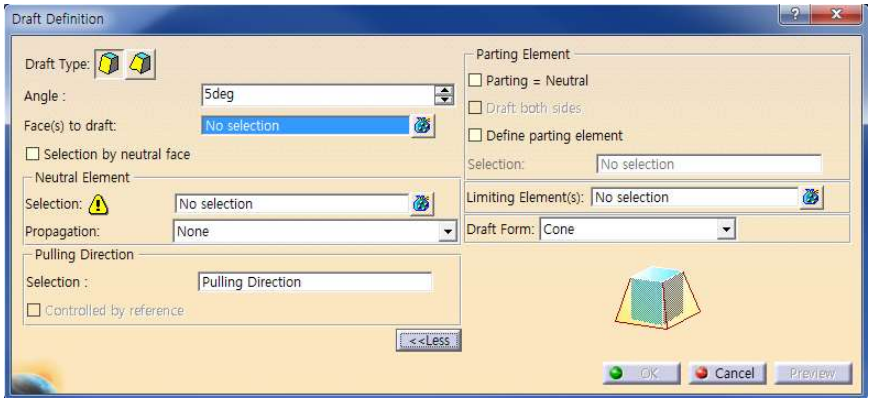


PART DESIGN TOOLS

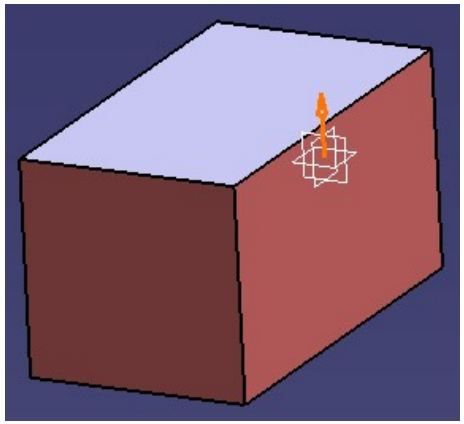
Dress-Up Features

Draft Angle 

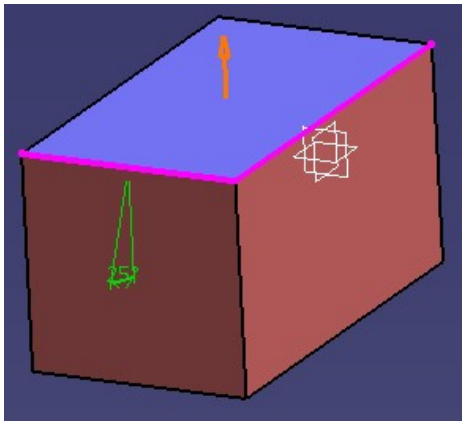
- 각도를 갖는 면 생성

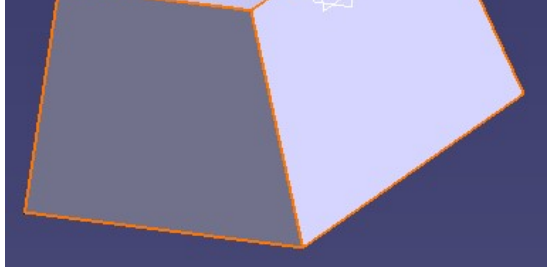


Draft 면 선택



Draft 방향, 각도 선택



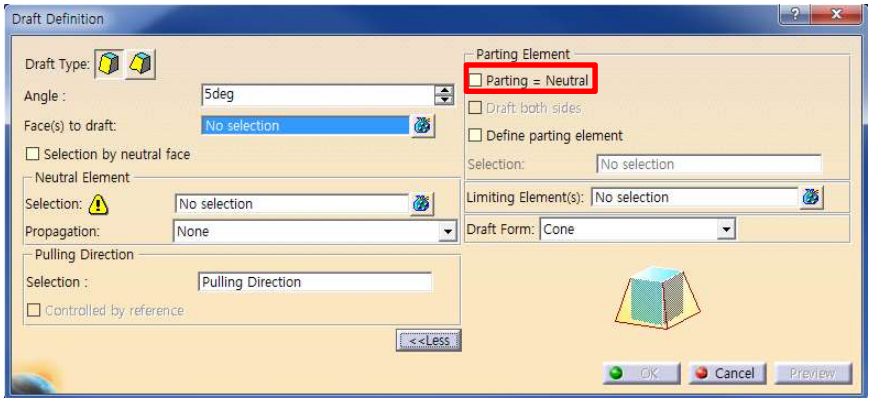


PART DESIGN TOOLS

Dress-Up Features

Draft Angle

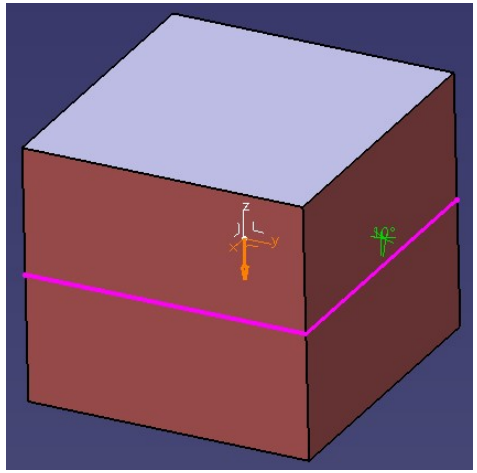
- 각도를 갖는 면 생성



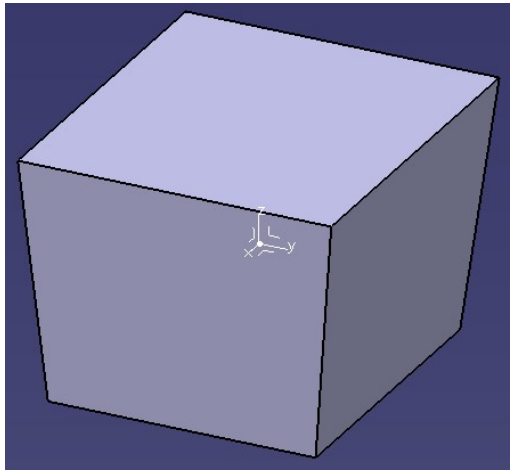
중심면(neutral element)으로 부터 선택한 방향에 속하는 면만을 draft 적용시킬 수 있다.

Parting=Neutral 클릭 후 Draft both sides를 이용하면 중심면을 기준으로 대칭인 draft를 적용히킬 수 있다.

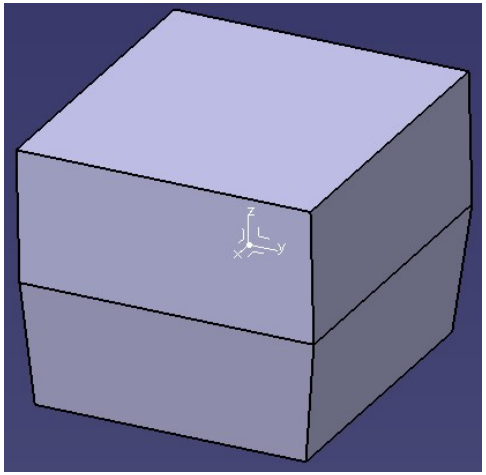
Draft 면 선택



<Without 'Parting=Neutral'>



<With 'Parting=Neutral'>

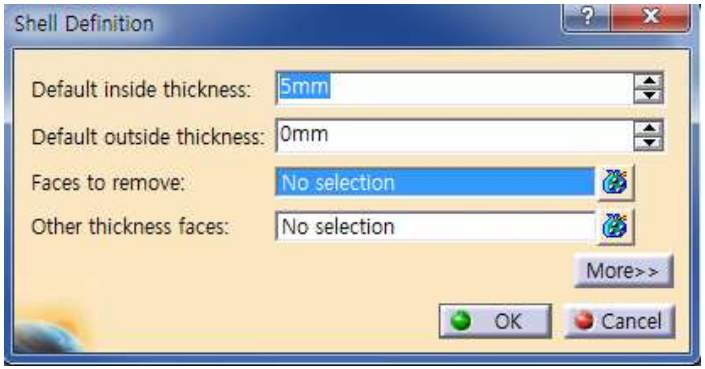


PART DESIGN TOOLS

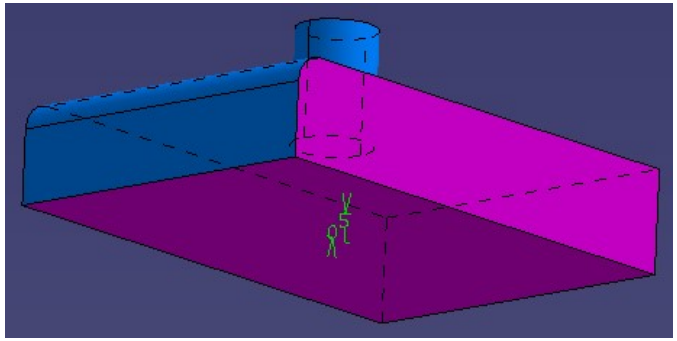
Dress-Up Features 

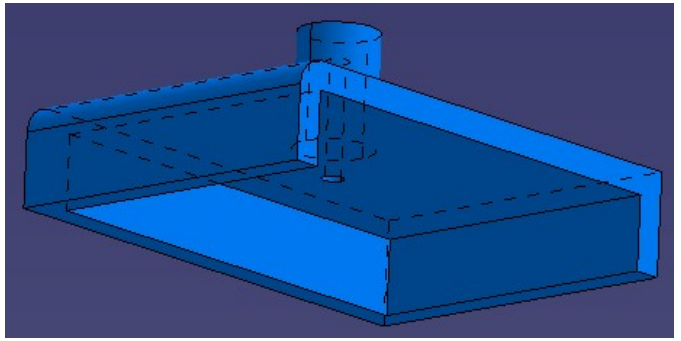
Shell 

- 주어진 두께를 남기고 나머지 부분 제거

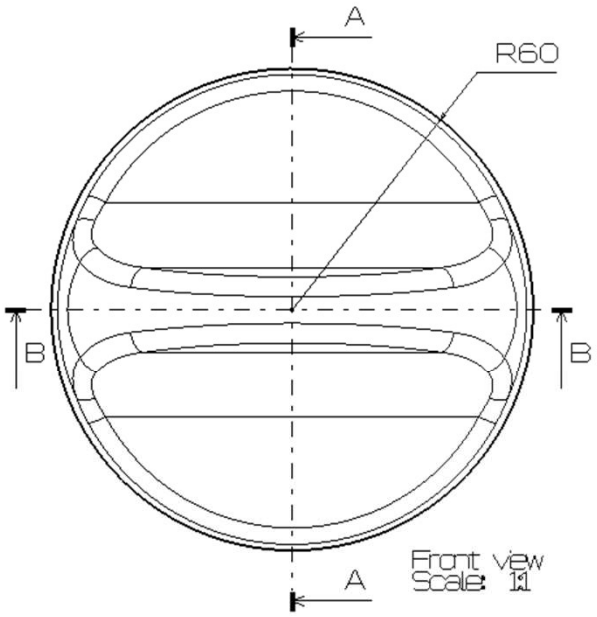


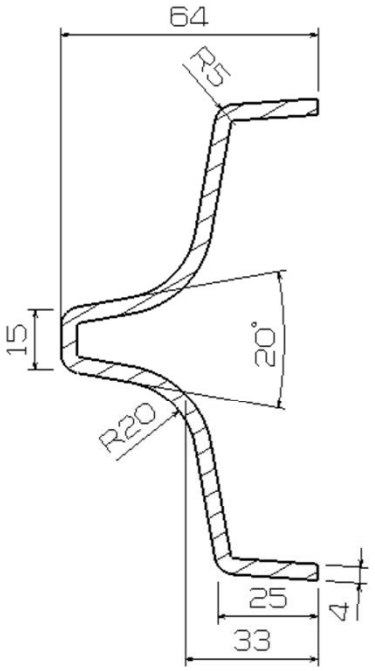
제거할 면 선택

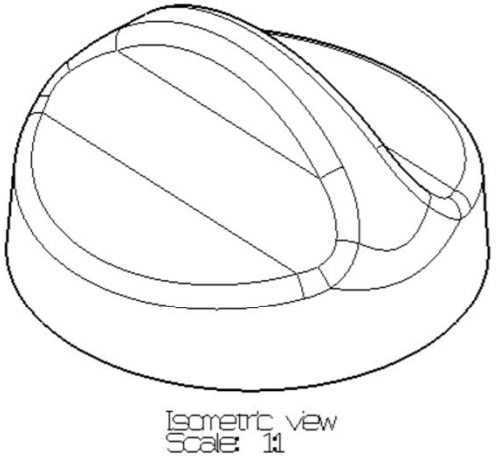


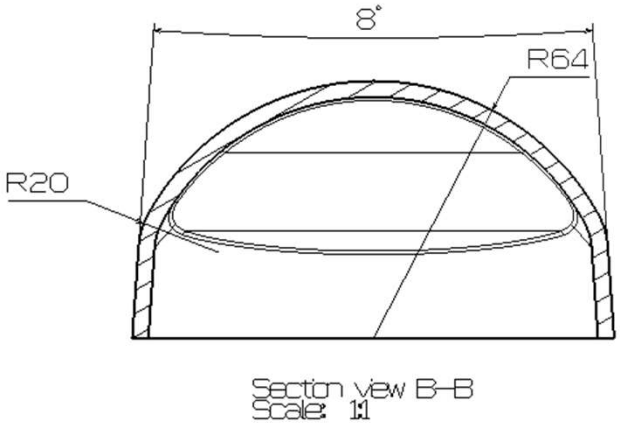


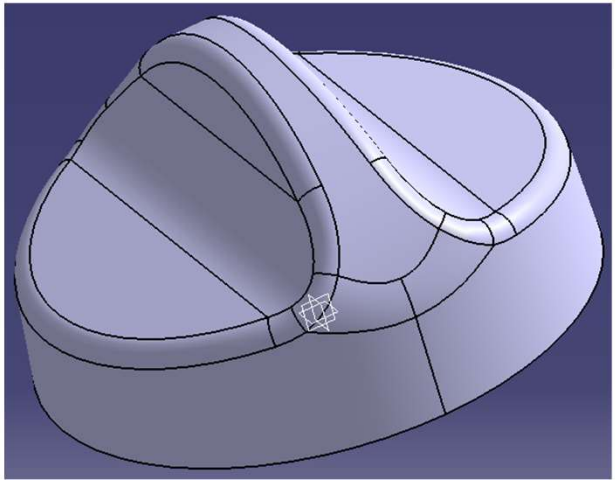
실습 예제









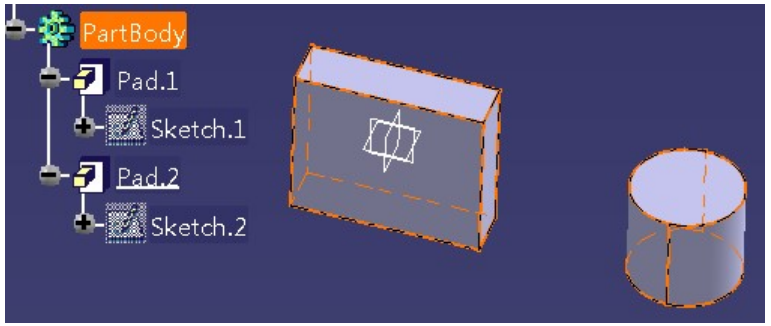


CONTENTS

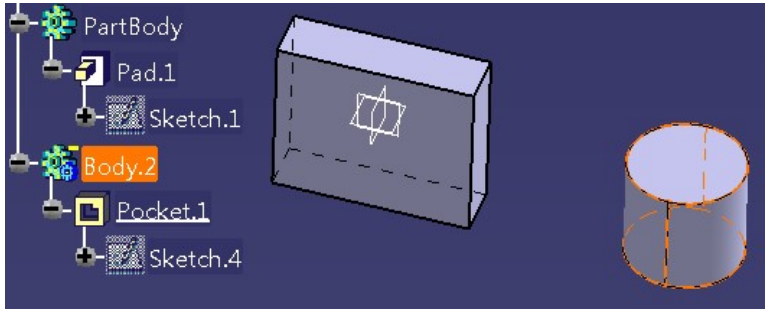
- ✓ 시작하기
- ✓ Reference Elements
- ✓ Sketch-Based Features
- ✓ Dress-Up Features
- ✓ **Transformation Features**
- ✓ Boolean Operation

Multi-Body

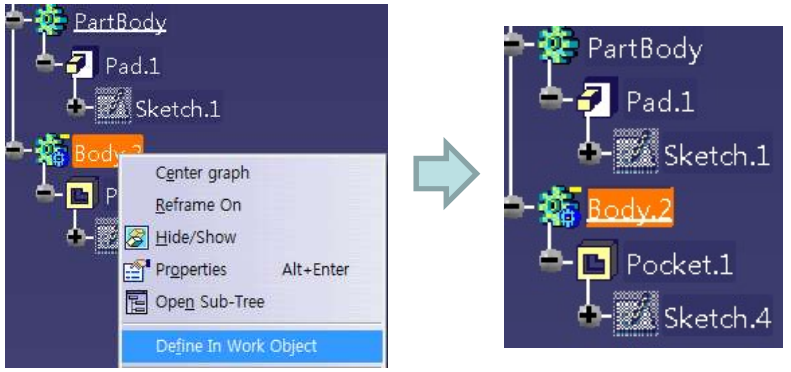
- Part Design을 하게 되면 기본적으로 PartBody에서 시작을 하게 되는데, 하나의 Body에서 작업을 진행할 경우 오른쪽 그림과 같이 분리된 물체도 하나의 모델로 인식하게 된다.



- 이를 방지하기 위해 메뉴의 Insert에서 Body를 생성하면 따로 작업을 할 수 있게 된다.
- Body를 나누어 작업할 경우 Pocket 같이 제거하는 기능도 오른쪽 작업트리와 같이 음의 부호를 갖는 모델로 표현 할 수 있다.



- Body를 여러 개 사용할 경우 작업이 나뉘어 지기 때문에 오른쪽 그림과 같이 작업하고자 하는 Body에서 마우스 오른쪽 버튼을 누르고 "Define In Work Object" 를 클릭하여 활성화 시켜주어야 한다. 활성화가 되면 해당 Body에 흰색 줄이 생성된다.

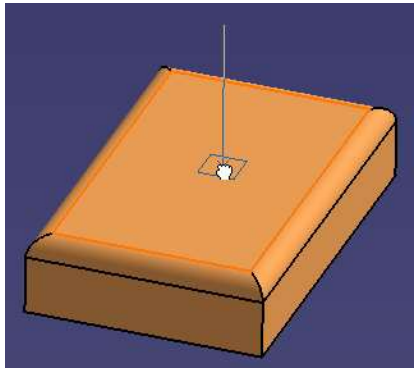


PART DESIGN TOOLS

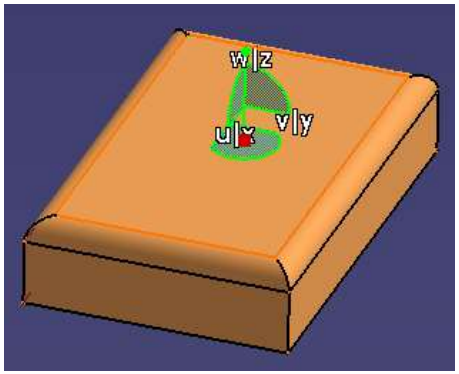
Compass 

- CATIA 화면 상단 우측에 위치

Compass를 드래그함



초록색 Compass 위치



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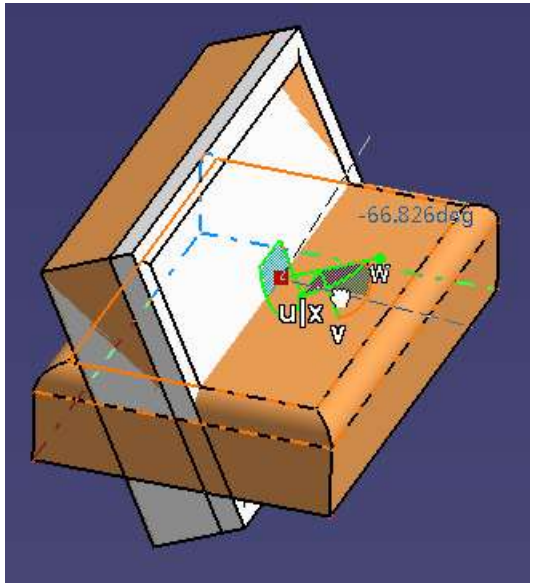
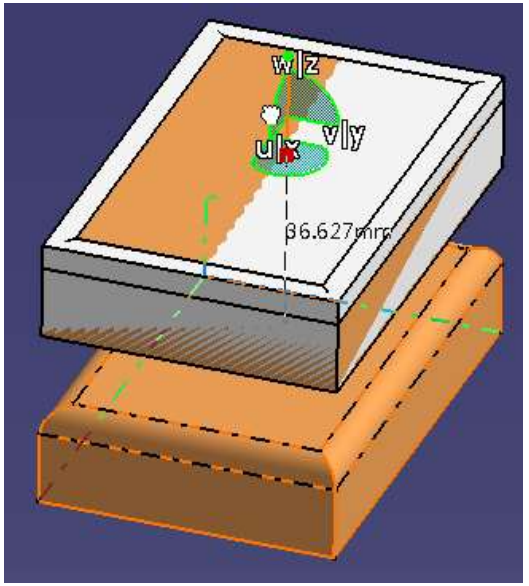
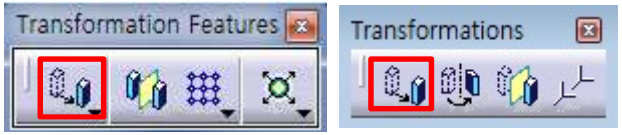


<이동>



PART DESIGN TOOLS

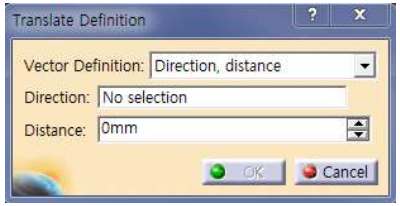
Transformation Features

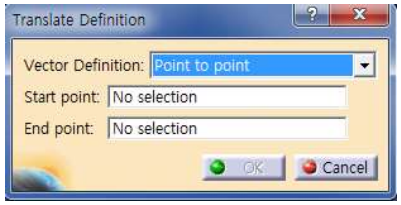


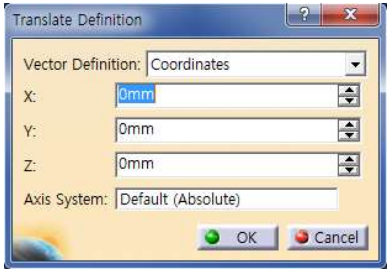
Translation 

- 모델을 위치 이동 시킴

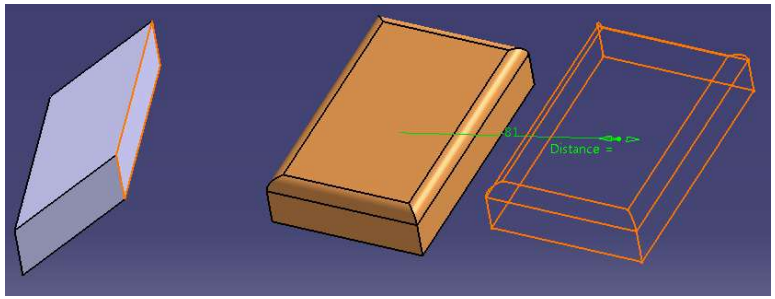
Translation types



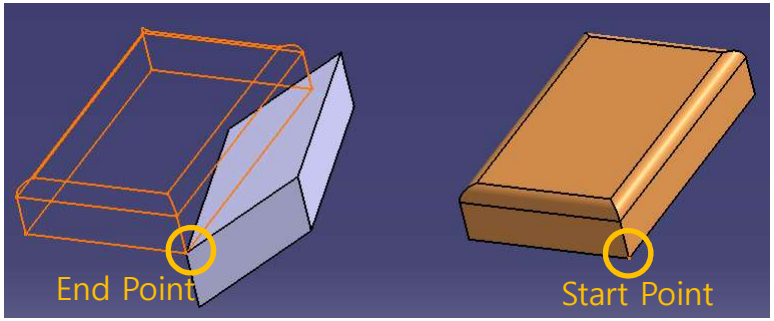




<Direction, distance>

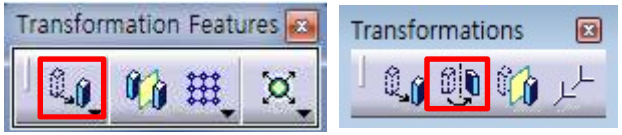


<Point to point>



PART DESIGN TOOLS

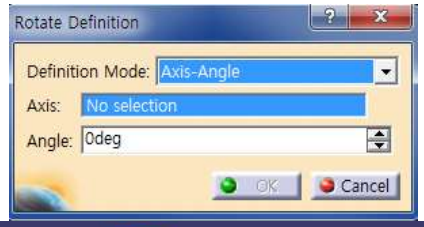
Transformation Features

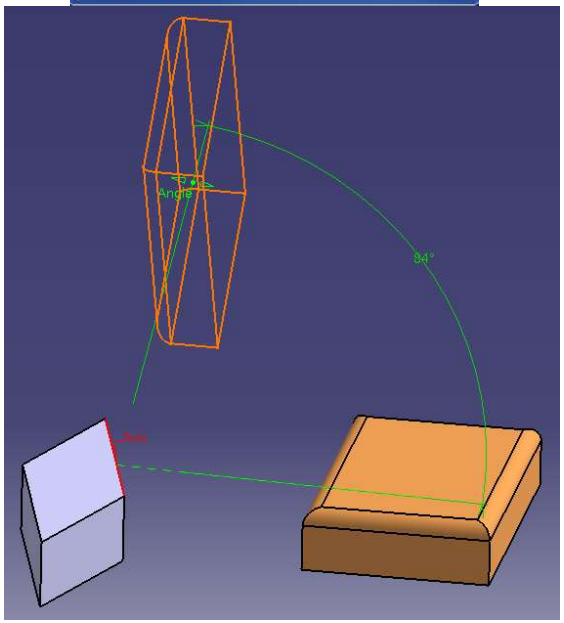


Rotation 

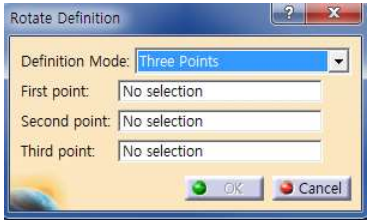
- 모델을 회전 이동 시킴

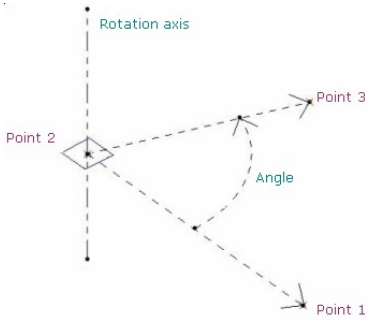
Axis-Angle

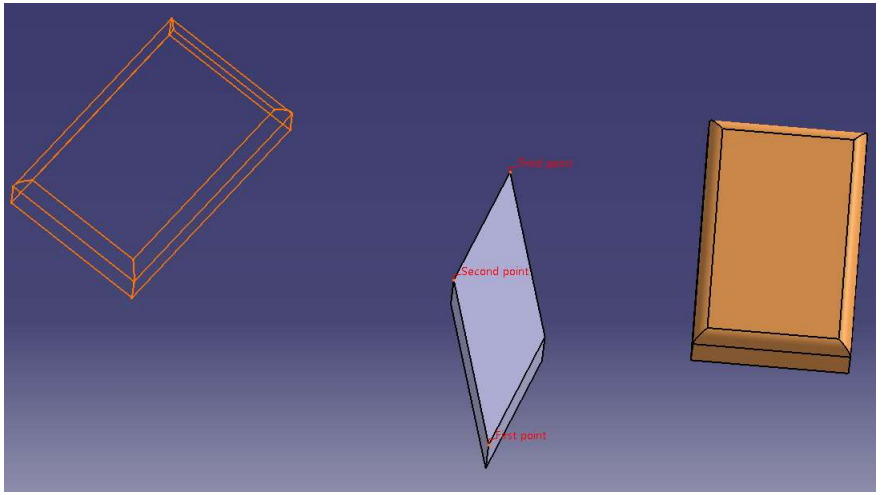




Three Points-Angle





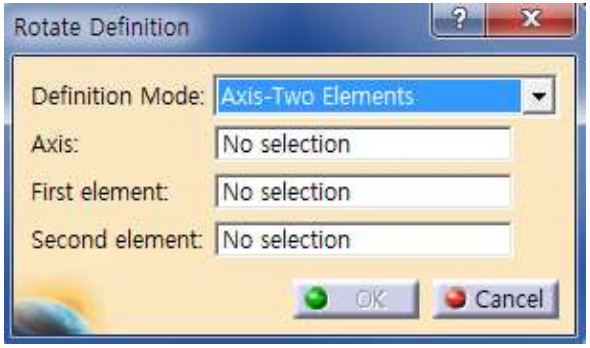


PART DESIGN TOOLS

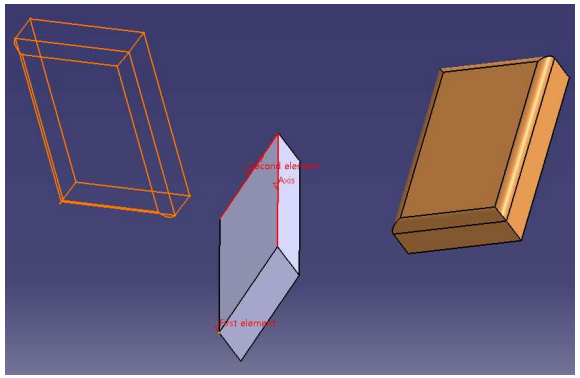
Transformation Features

Rotation

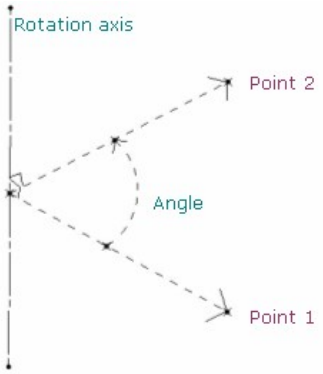
Axis-Two Elements



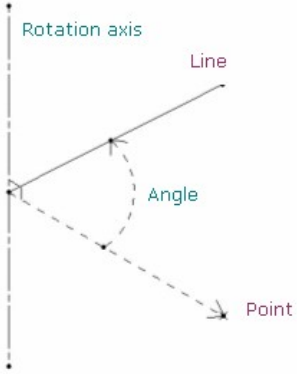
<Axis/line/point>



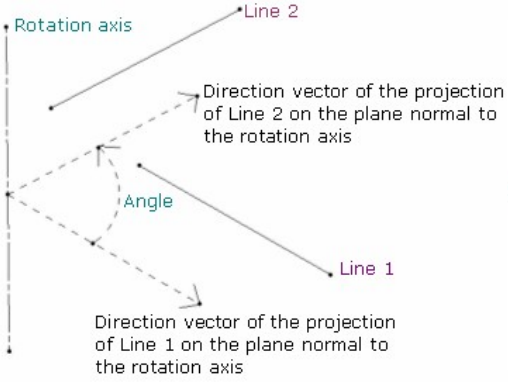
Axis/point/point



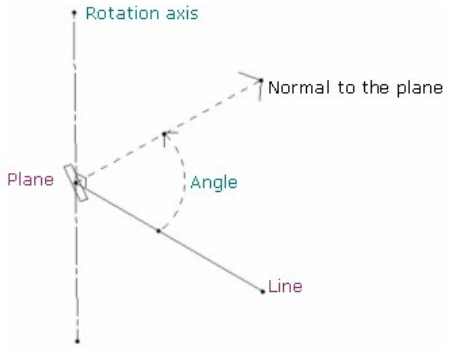
Axis/line/point



Axis/line/line

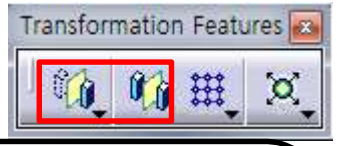


Axis/plane/line

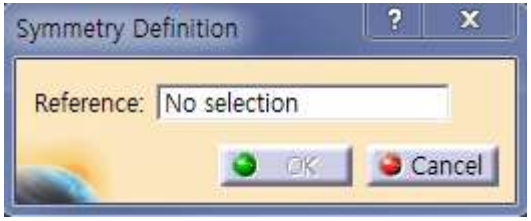


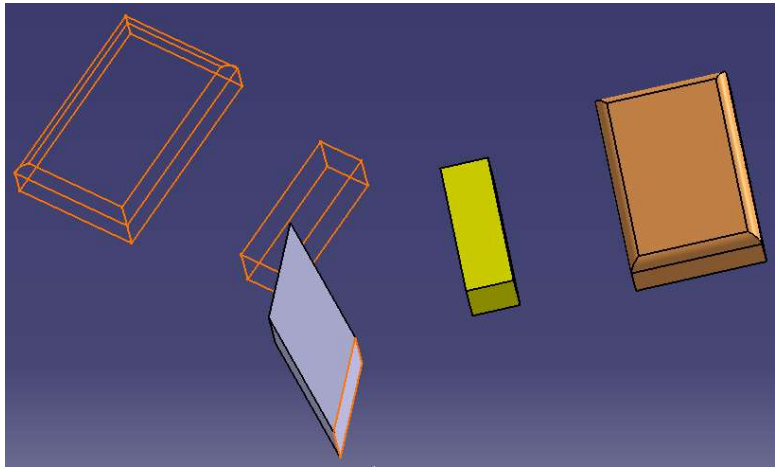
PART DESIGN TOOLS

Transformation Features

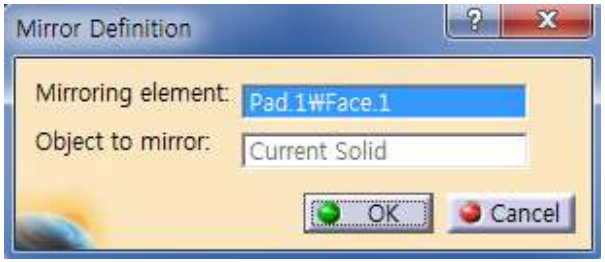


Symmetry 
- Reference에 대칭 이동





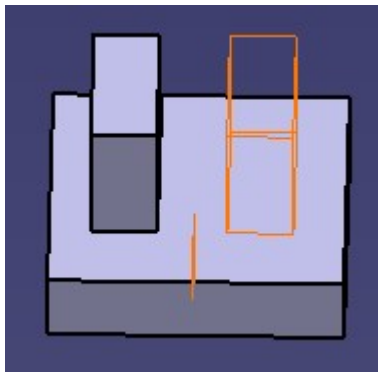
Mirror 
- 대칭 복사



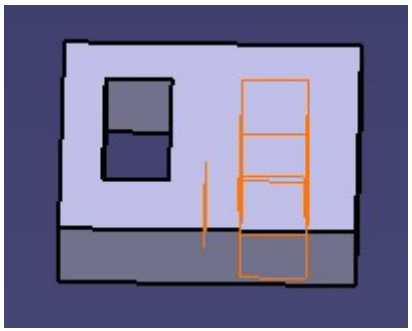
기준위치

대상

<Pad 대칭 복사>



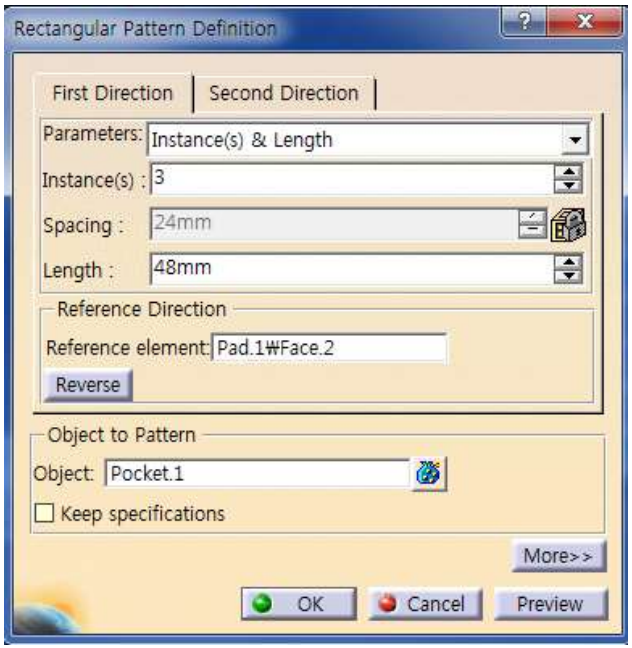
<Pocket 대칭 복사>



PART DESIGN TOOLS

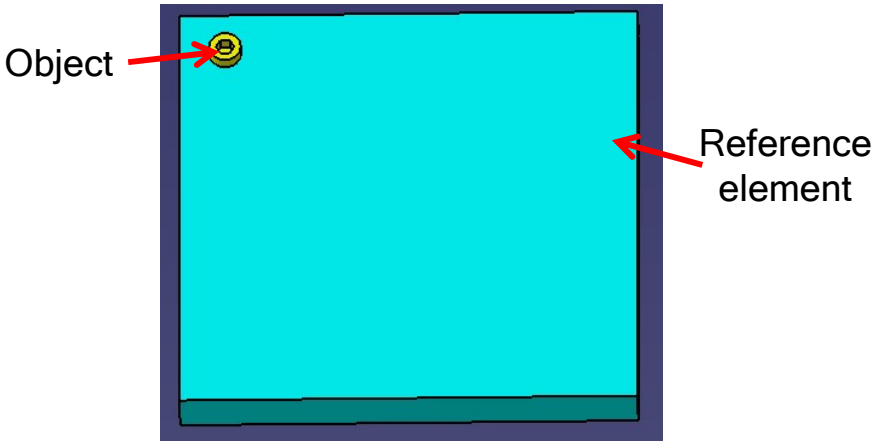
Transformation Features

Rectangular Pattern
- 수직방향의 패턴 생성

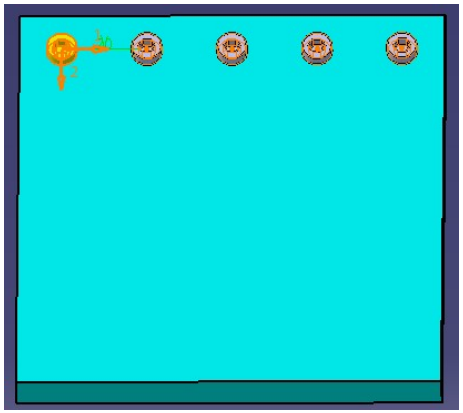


- Preview 클릭 시 생성되는 주황색 point 를 클릭하면 제거/재생성 가능
- 를 클릭하여 object 내 다른 기능들 (Pad, Fillet 등)도 함께 패턴 생성 가능

<기본 모델>



<First Direction>



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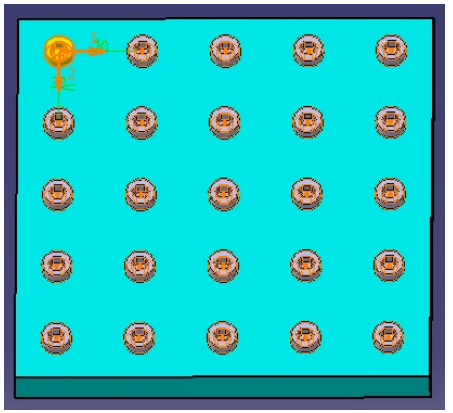
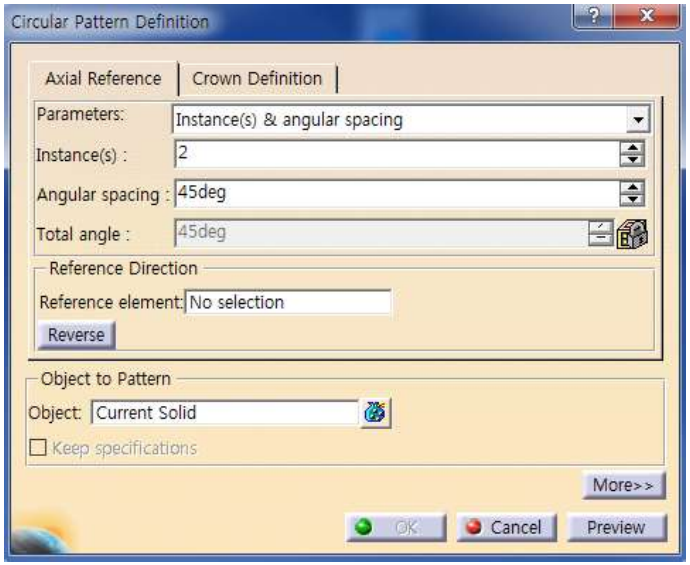


PART DESIGN TOOLS

Transformation Features

Circular Pattern
- 원형 패턴 생성



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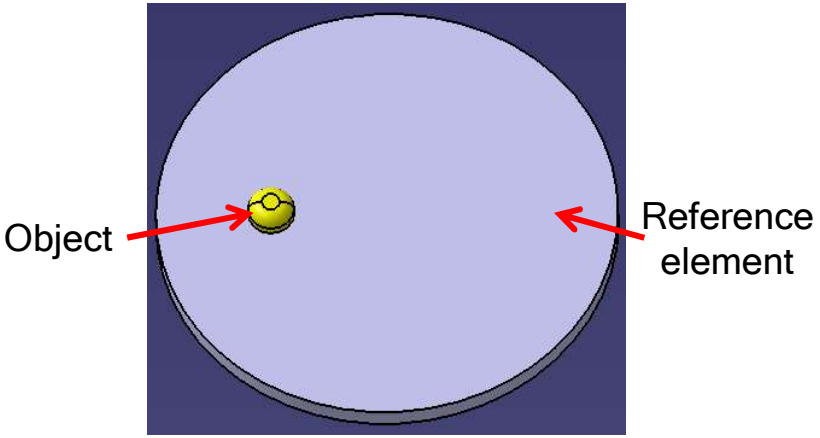

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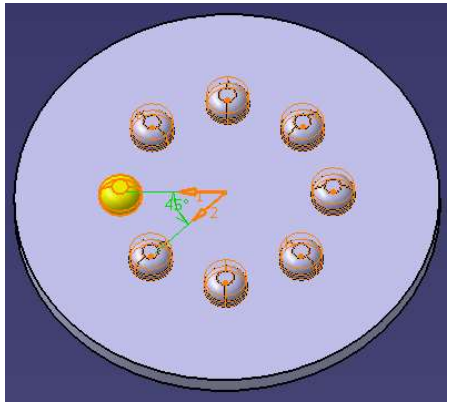

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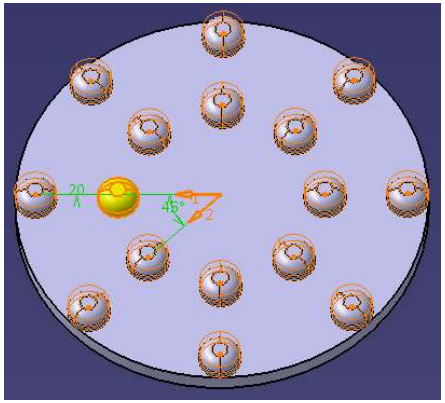
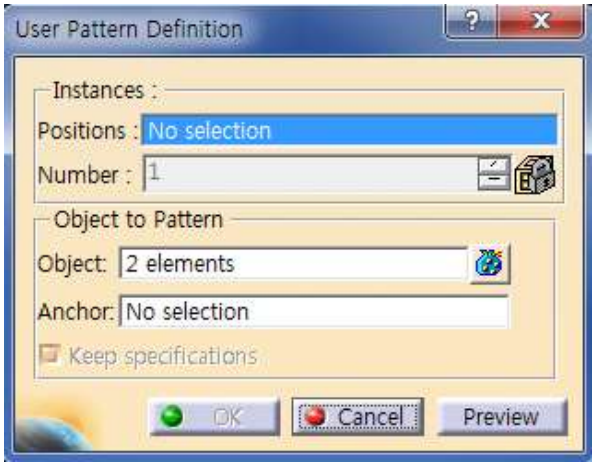


PART DESIGN TOOLS

Transformation Features

User Pattern
- 사용자 임의의 패턴 생성



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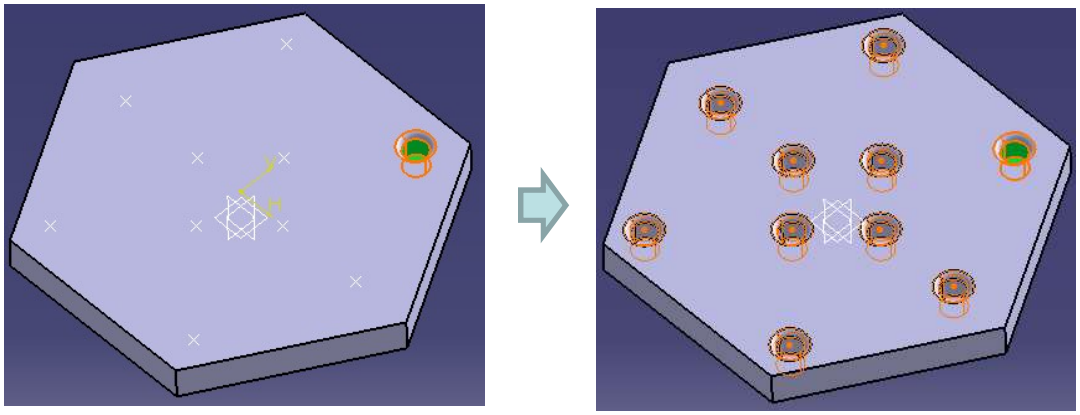
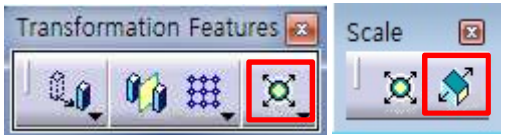


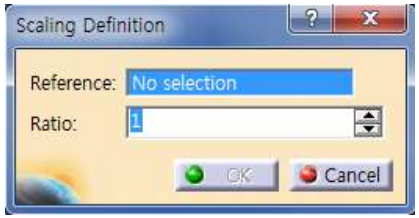
PART DESIGN TOOLS

Transformation Features

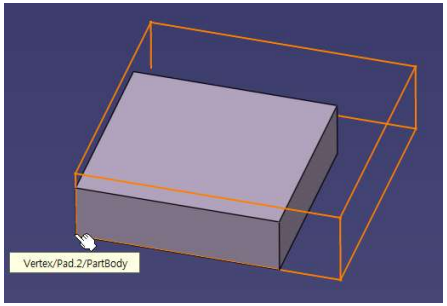


Scale

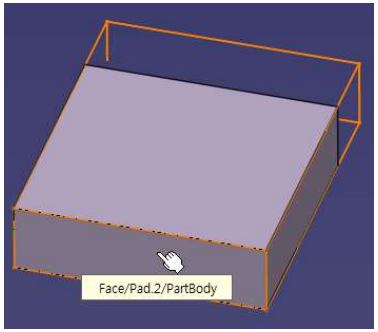
- 모델의 크기를 reference 기준으로 변경



<vertex 기준>

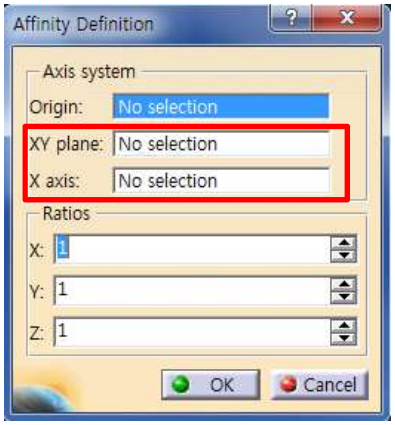


<plane 기준>



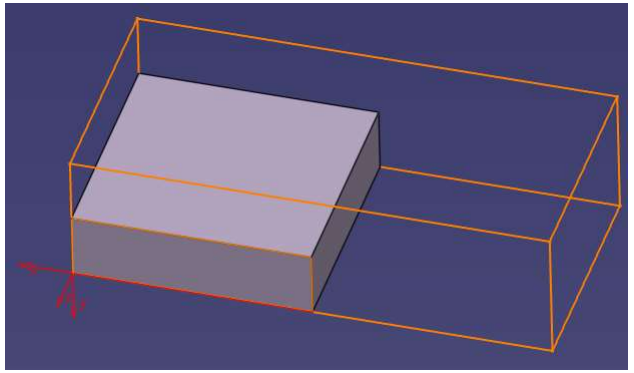
Affinity

- 선택 점을 기준으로 방향에 따라 크기 변경



설정하지 않을 경우
compass를 기준으로 진행

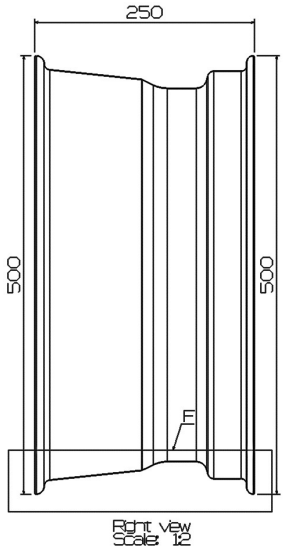
<X=2, Y=2, Z=1>

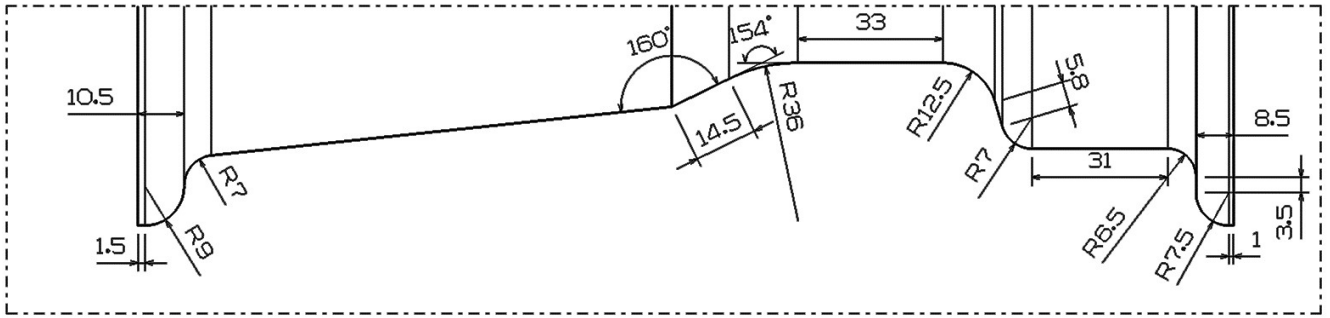


실습 예제

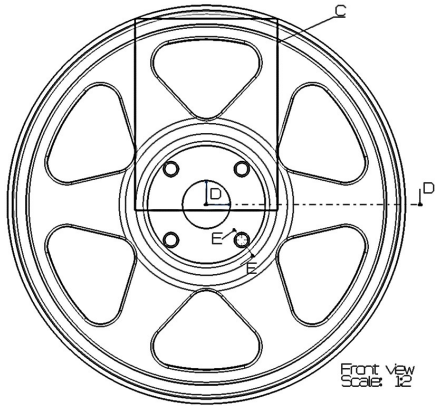
Automobile Wheel

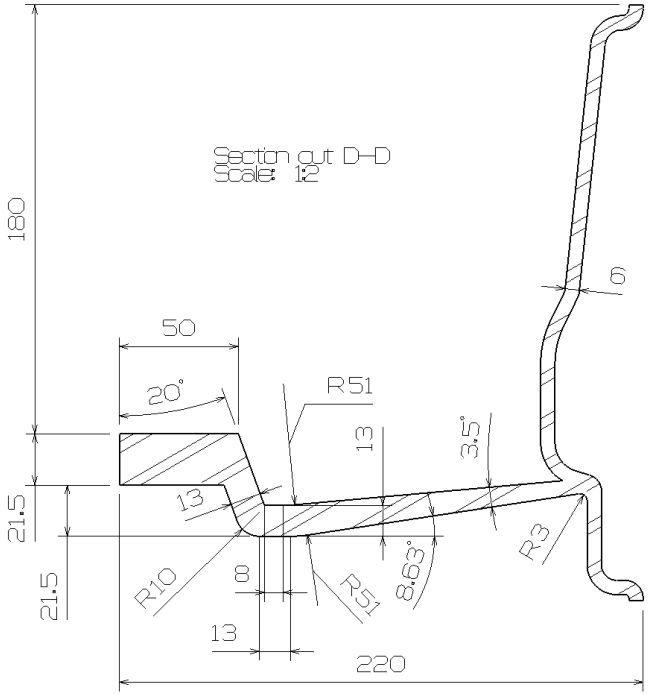
- 표기하지 않은 fillet 값은 3mm
- 표기하지 않은 line 연결은 tangency constraint 적용

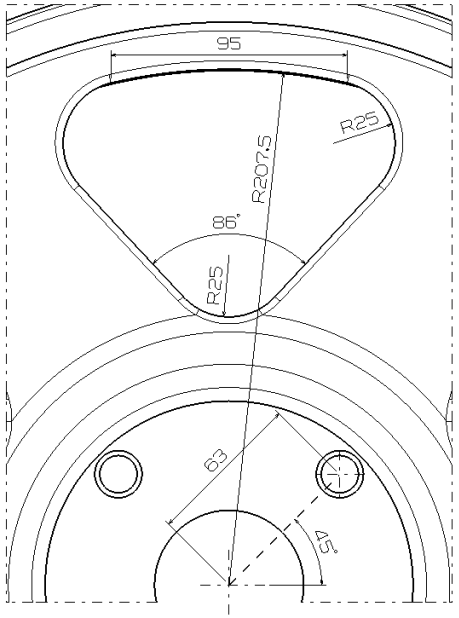




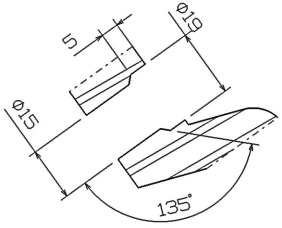
Detal F
Scale: 1:1







Detal C
Scale: 1:1



Section cut E-E
Scale: 1:1

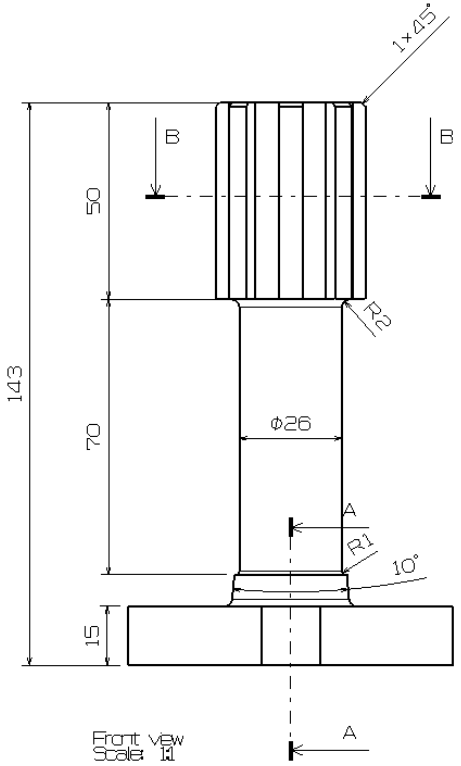
CATIA v5 모델링 예제 가이드, 최홍태, 미전사이언스, 2011

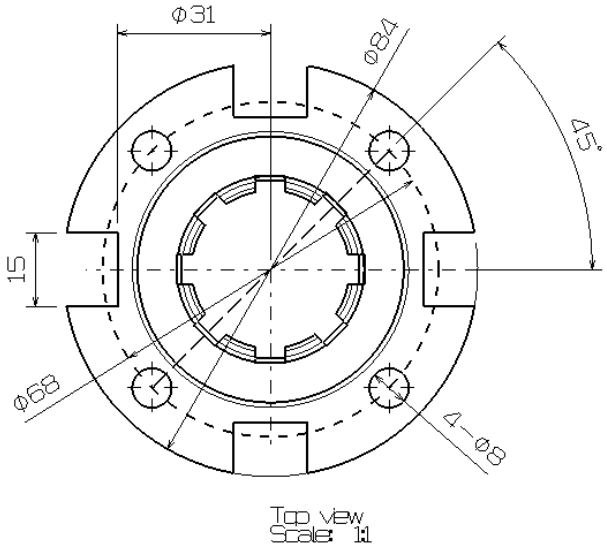
AUTOMOBILE WHEEL

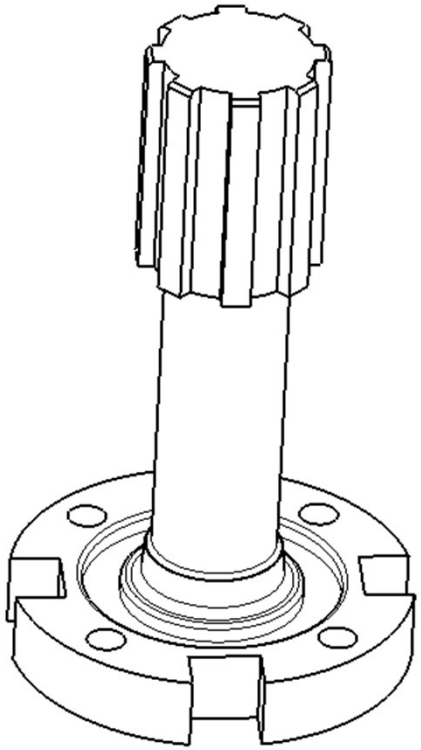
Shading View

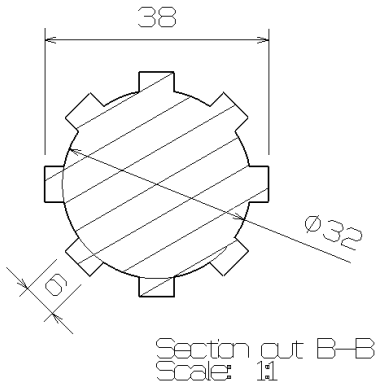
Shading with Material View(Aluminum)

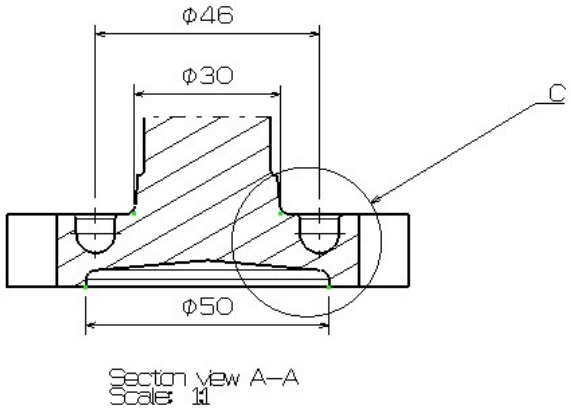
실습 과제

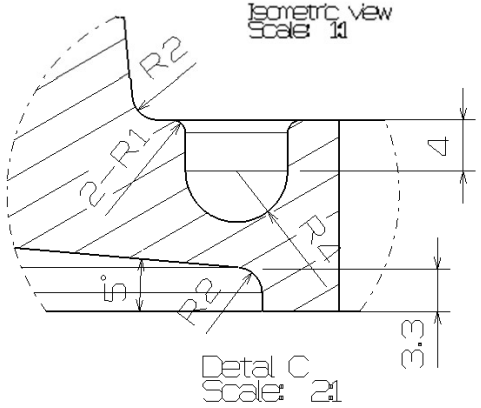












CONTENTS

- ✓ 시작하기
- ✓ Reference Elements
- ✓ Sketch-Based Features
- ✓ Dress-Up Features
- ✓ Transformation Features
- ✓ **Boolean Operation**

PART DESIGN TOOLS

Boolean Operations 

Assemble : Body의 속성을 유지한 채 Body간 결합을 하는 기능

Add : Body의 속성을 무시하고 형상만으로 Body간 결합을 하는 기능

Remove : Body간 제거를 하는 기능

Intersect : Body간 교차되는 부분을 남겨두고 나머지 부분을 제거 하는 기능

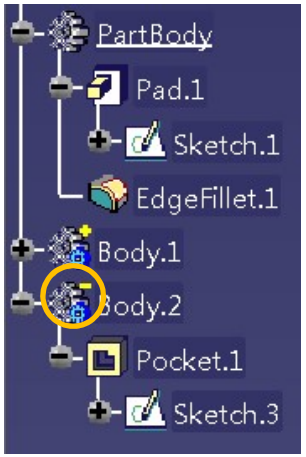
Union Trim : Body간에 사용자가 필요한 부분만 남겨두고 나머지는 제거하는 기능

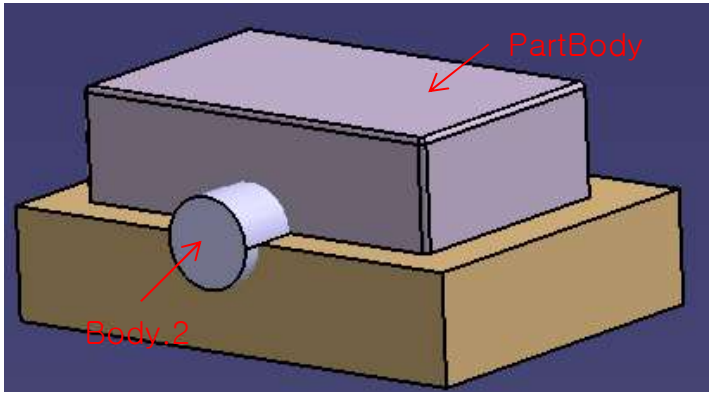
Remove Lump : 하나의 Body에서 완전히 분리되어진 부분을 제거 하거나 빈 공간으로 남겨진 부분을 제거하는 기능

PART DESIGN TOOLS

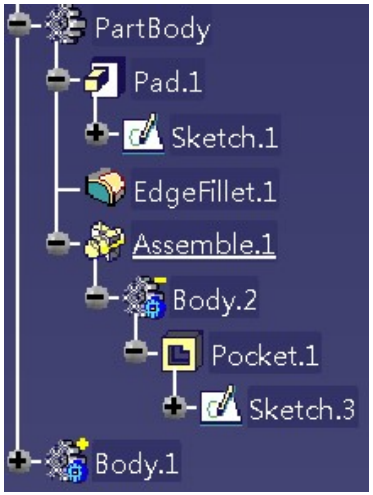
Boolean Operations

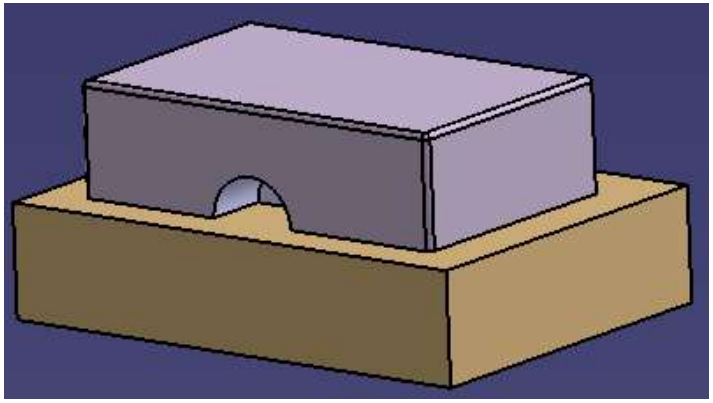
Assemble





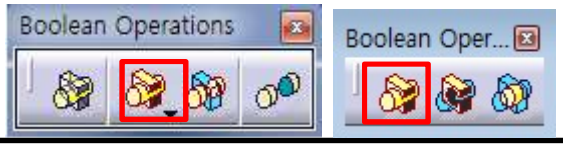
Body2를 Assemble



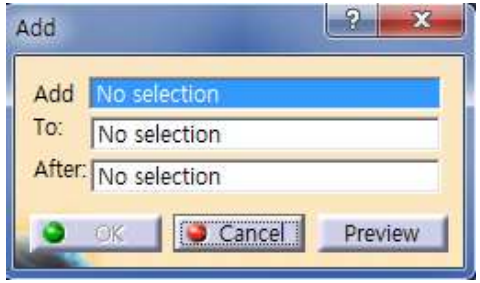


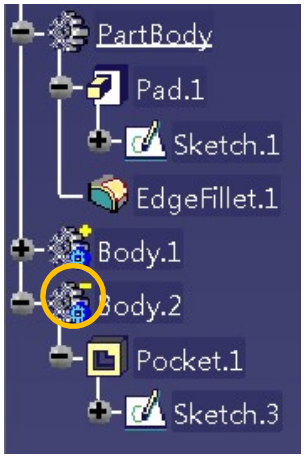
PART DESIGN TOOLS

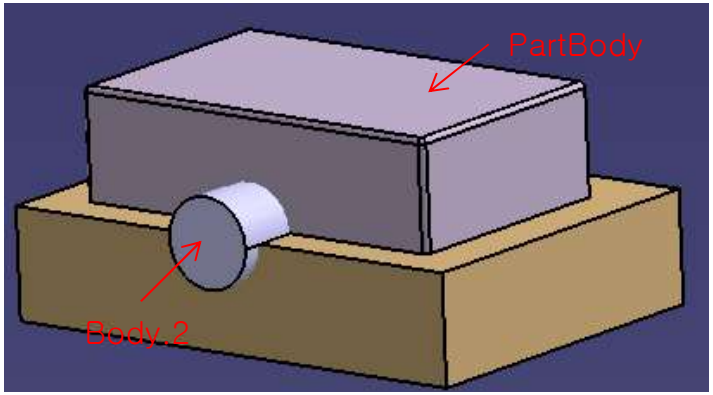
Boolean Operations



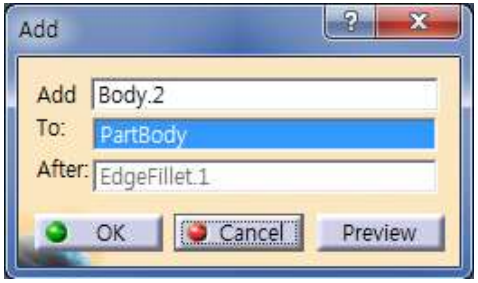
Add 

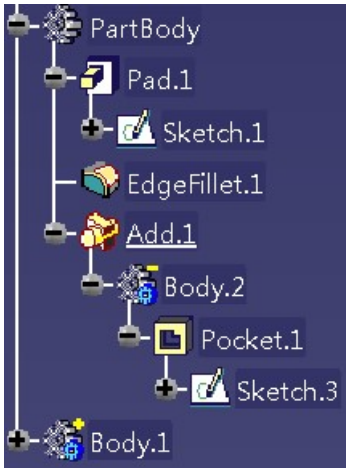


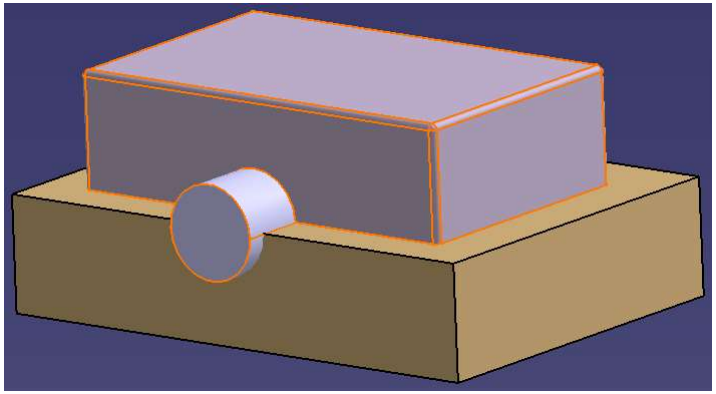




Body2를 Add

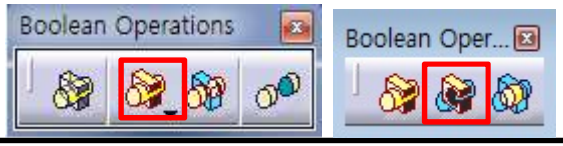




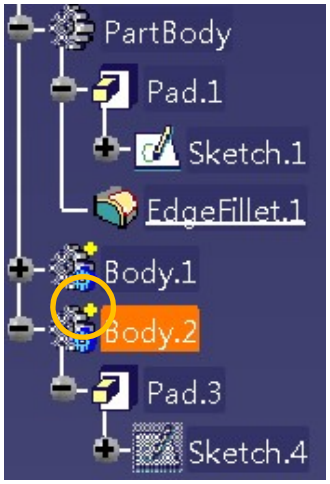


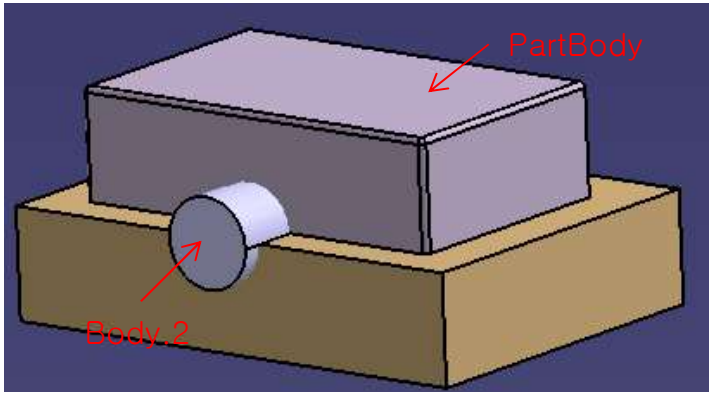
PART DESIGN TOOLS

Boolean Operations

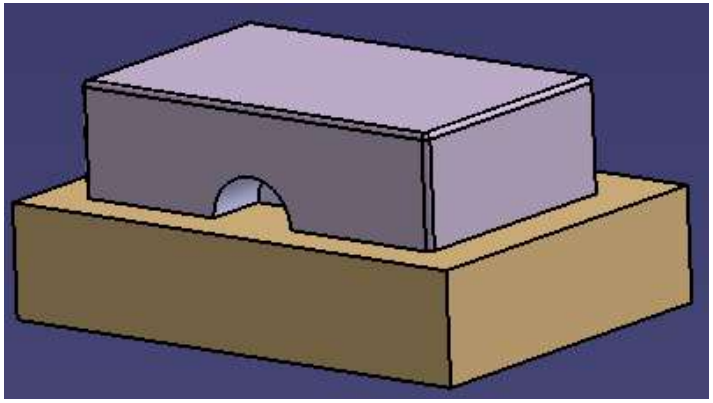


Remove



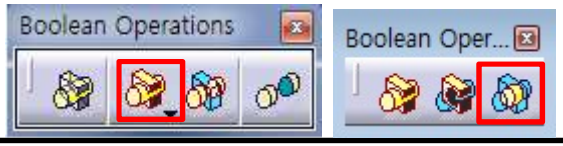


Body2를 Remove

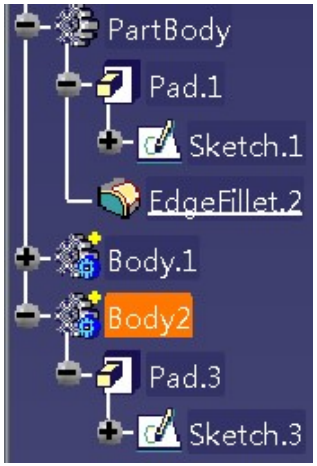


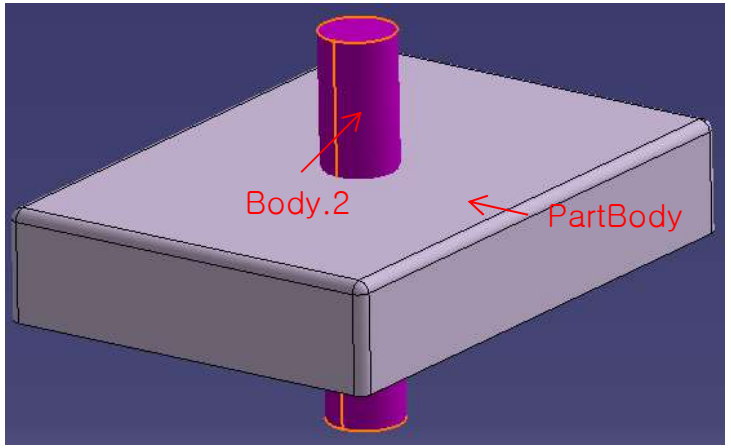
PART DESIGN TOOLS

Boolean Operations



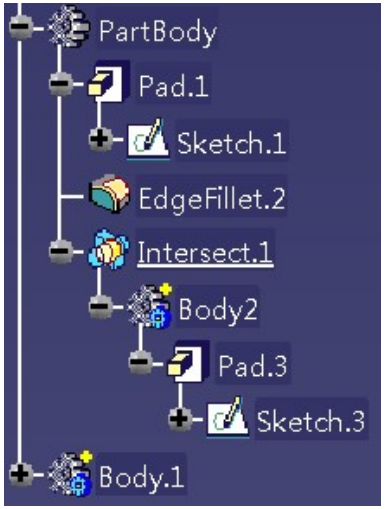
Intersect

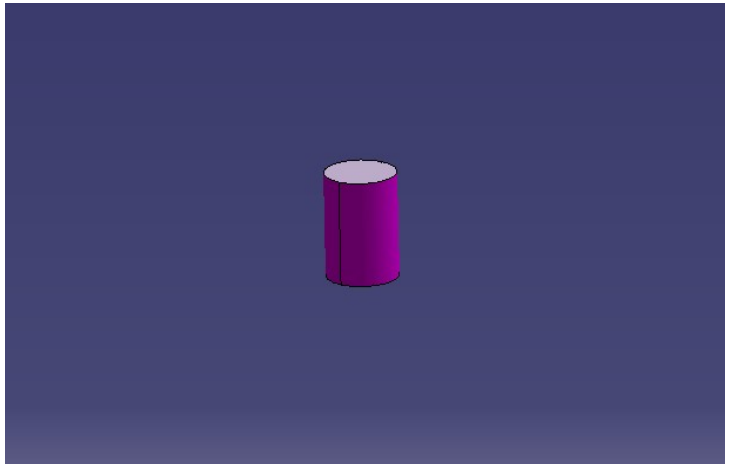




Body2를 Intersect

Body의 부호에 영향을 받지 않음

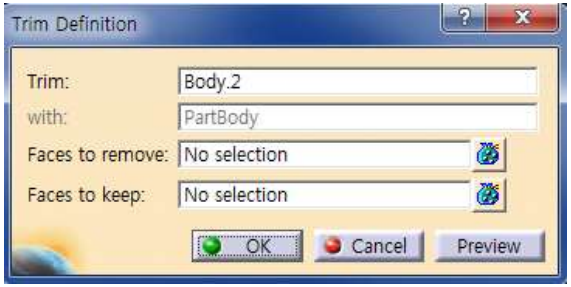


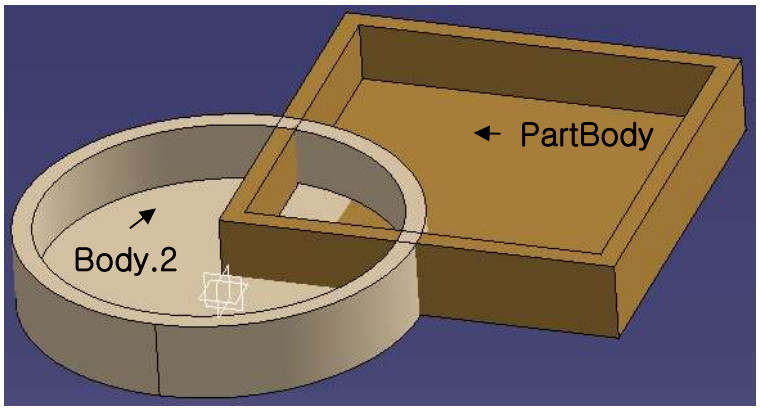


PART DESIGN TOOLS

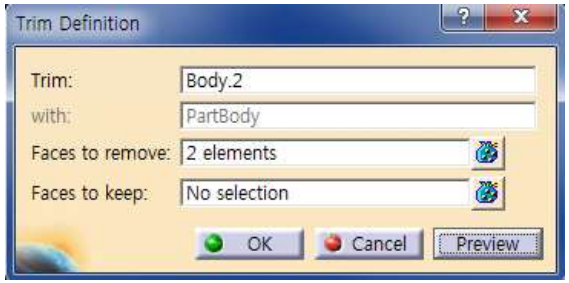
Boolean Operations

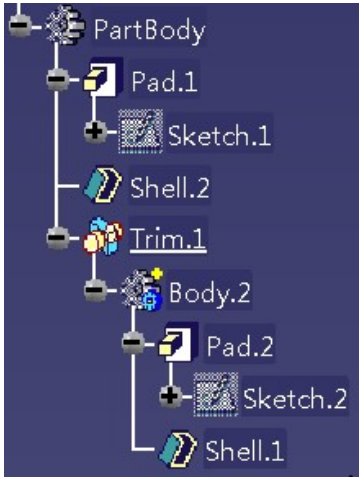
Union Trim

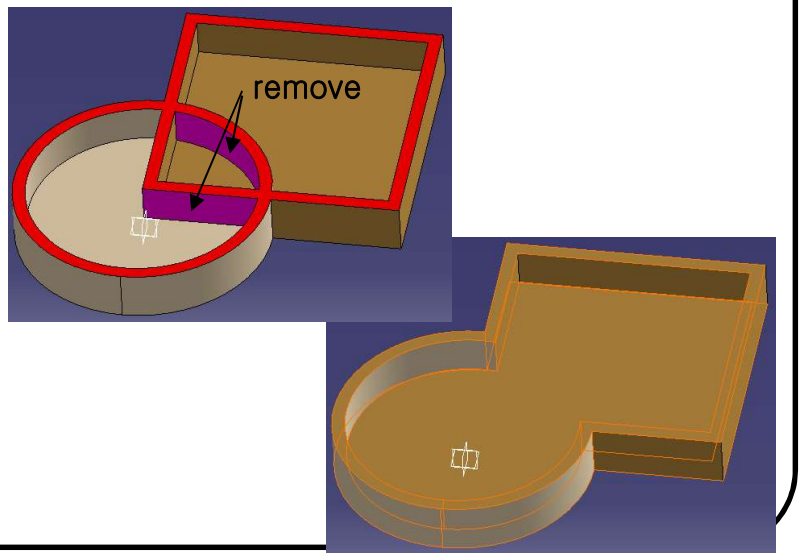




Body2를 Trim



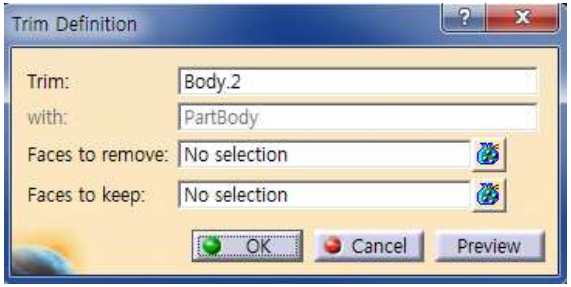


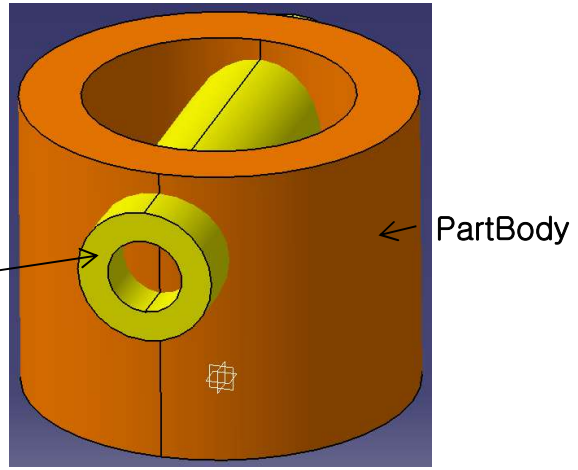


PART DESIGN TOOLS

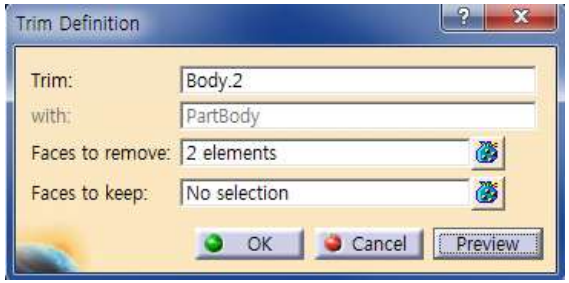
Boolean Operations

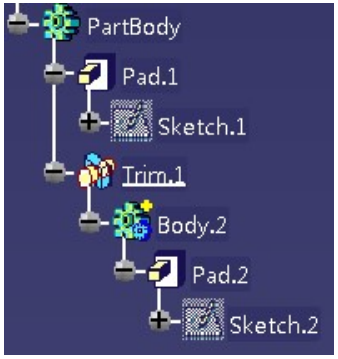
Union Trim

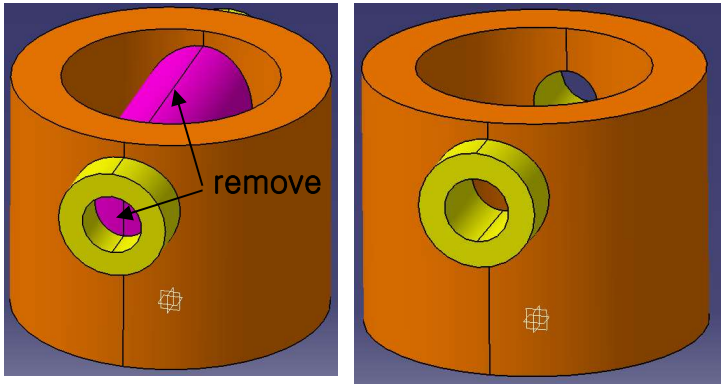




Body2를 Trim





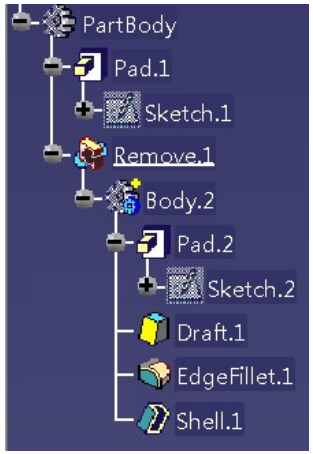


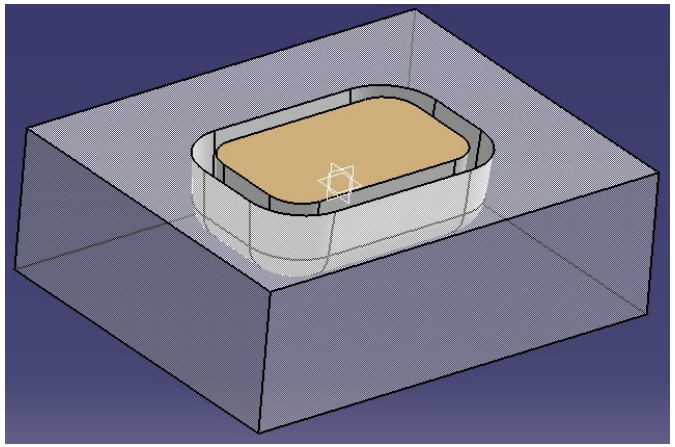
PART DESIGN TOOLS

Boolean Operations

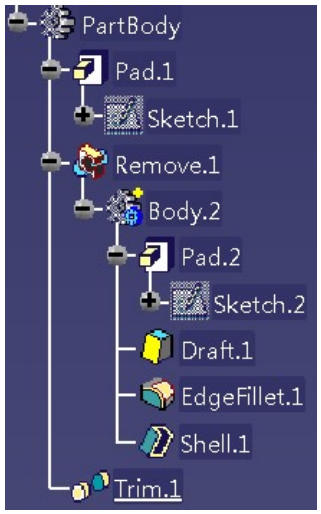
Remove Lump

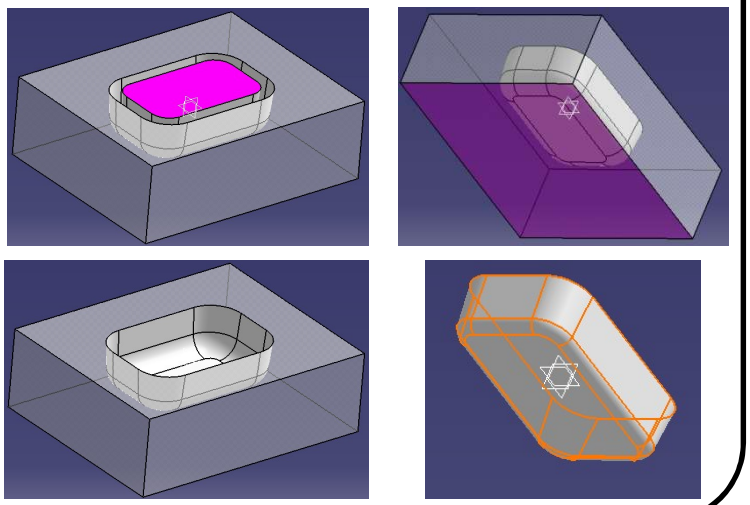
- Boolean operation 작업 후 불필요한 부분 제거





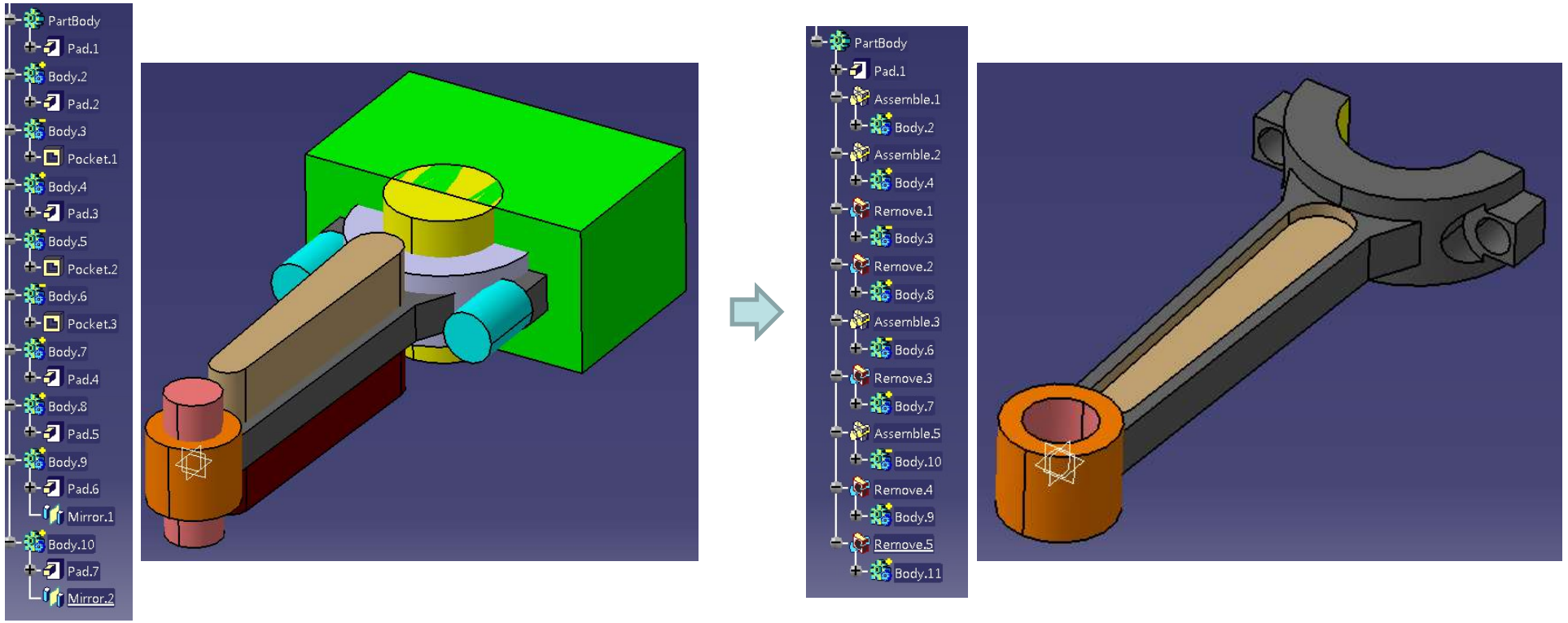
선택한 face를 Trim





실습 예제

Boolean Operation을 이용하여 Connecting Rod 모델링



실습 과제

Body를 나누어 진행한 뒤 Boolean Operation 이용