

# CATIA V5 교안



2019. 03.

기계과 조 남 철 교수

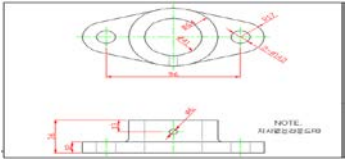
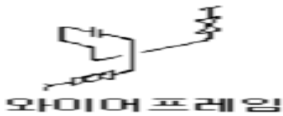

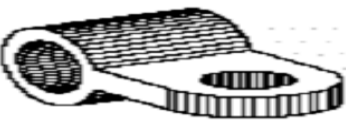
# I. 3D CAD 의 개요

## 1. CAD 의 개요 및 관련 용어

CAM (Computer Aided Manufacturing)	<b>컴퓨터 응용 제품생산</b>
CAE (Computer Aided Engineering)	<b>컴퓨터 응용 제품 해석 및 시뮬레이션</b>
CIM (Computer Integrated Manufacturing)	<b>CAD/CAM/CAE 의 통합 업무 시스템</b>
FA (Factory Automation)	<b>공장전체의 자동화</b>
PDM (Product Data Management)	제품 데이터 관리 시스템
PLM (Product Lifecycle Management)	제품 수명 전 기간 관리시스템 (기획, 개념, 설계, 생산, 폐기등)

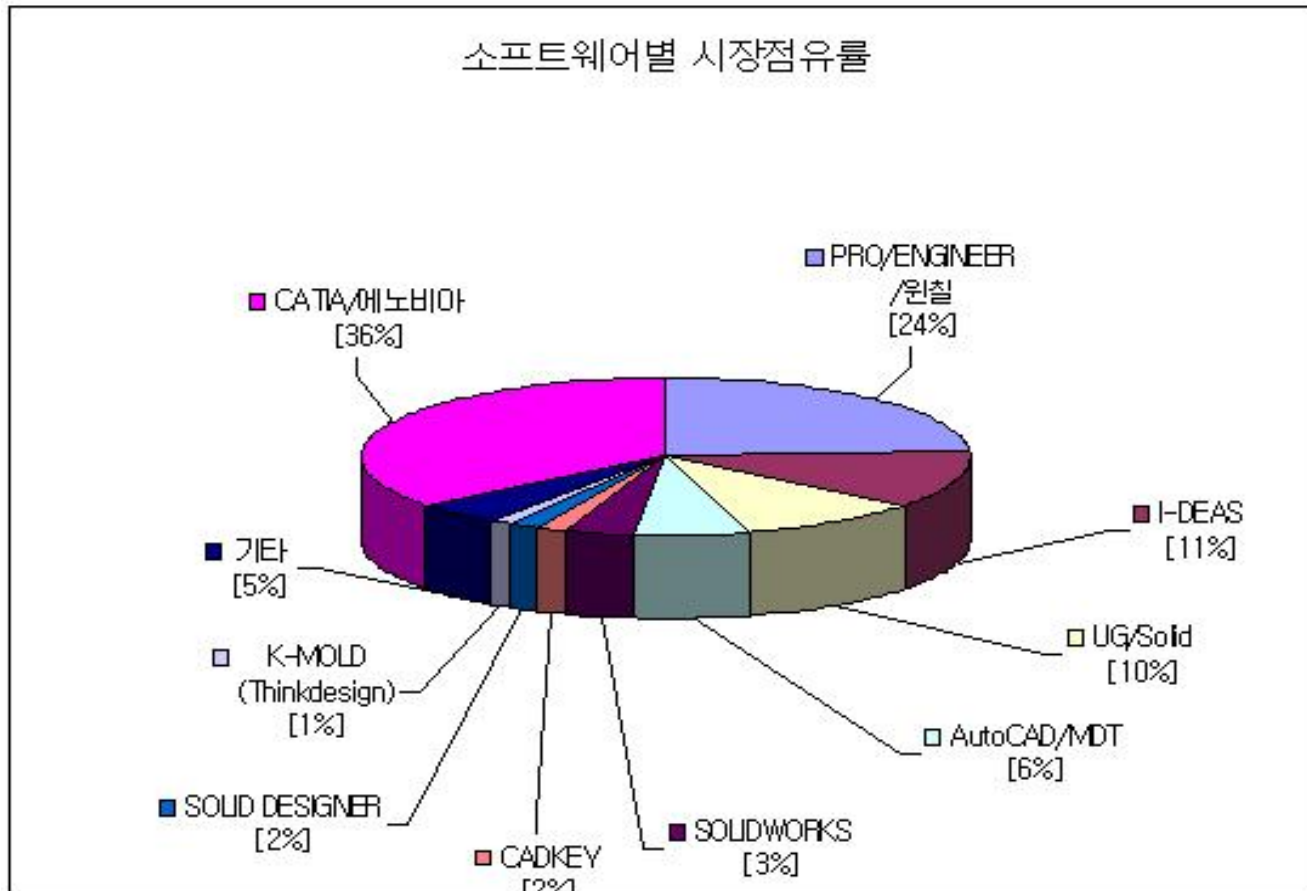
# I. 3D CAD 의 개요

## 2. CAD 모델의 표현

<p><b>2D Drafting</b></p>	
<p><b>3차원 와이어 프레임 모델 ( Wire-Frame Modeling )</b></p>	 <p>와이어프레임</p>
<p><b>서페이스 모델 ( Surface Modeling )</b></p>	 <p>메쉬</p>
<p><b>솔리드 모델 ( Solid Modeling )</b></p>	 <p>솔리드</p>

# I. 3D CAD 의 개요

## 3. 각종 CAD Software

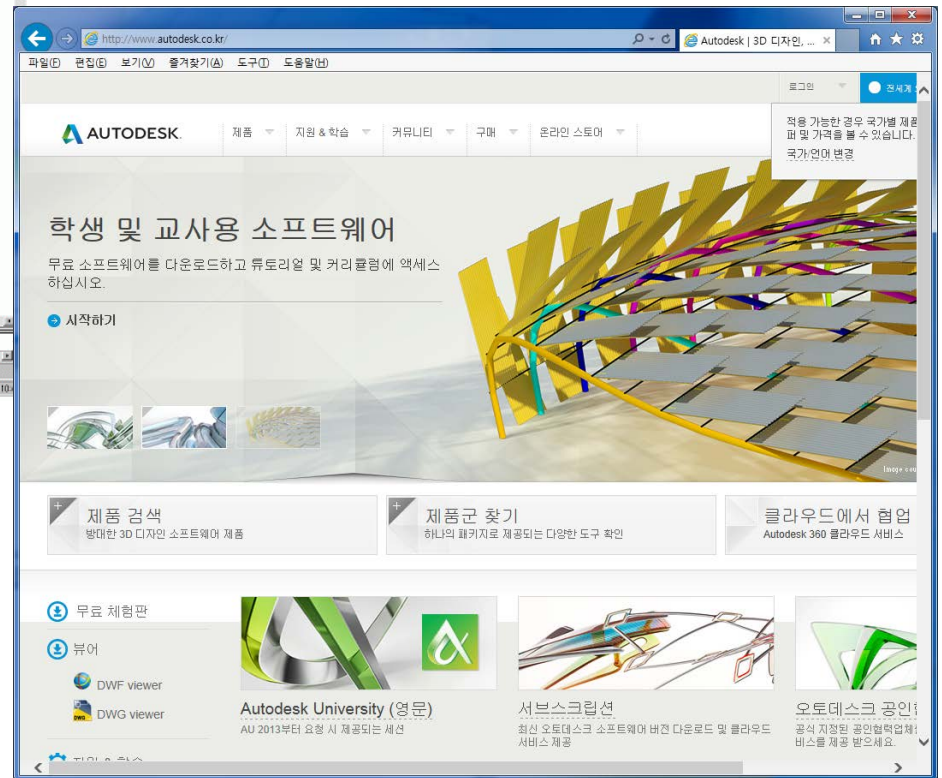
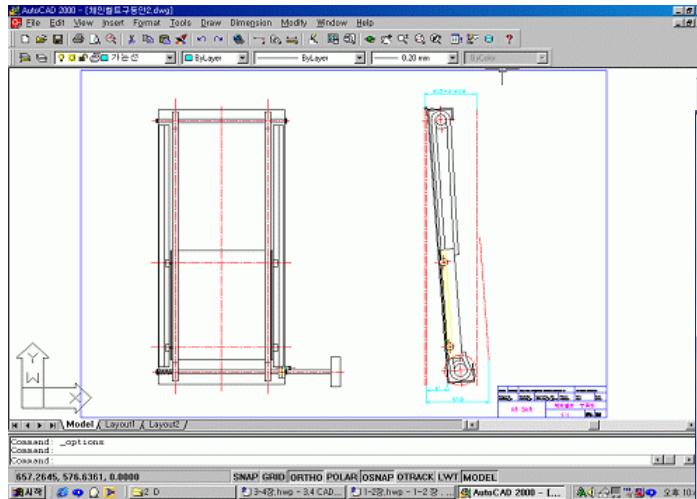




# I. 3D CAD 의 개요

## 3-1. Auto CAD

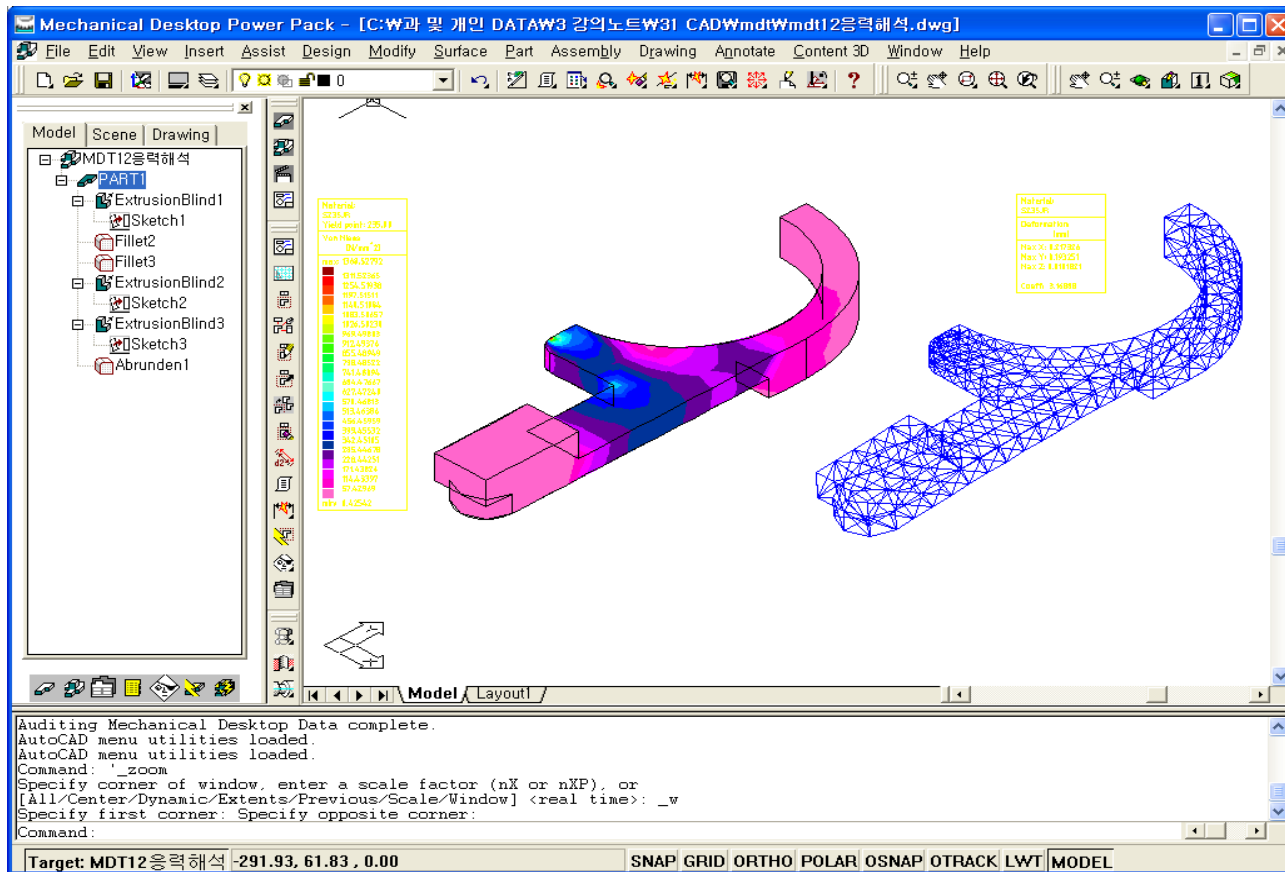
www.autodesk.co.kr



# I. 3D CAD 의 개요

## 3-2. MDT (1)

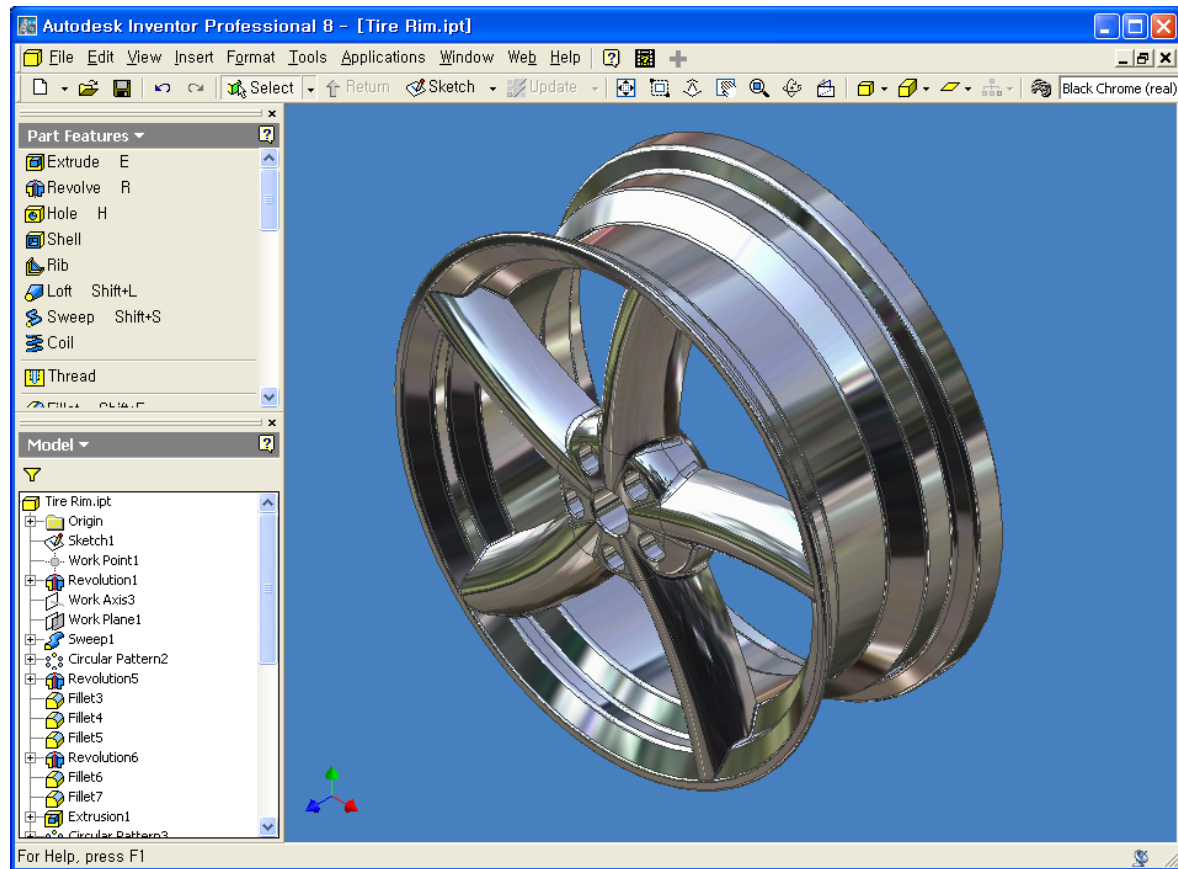
www.autodesk.com



# I. 3D CAD 의 개요

## 3-2. Inventor (2)

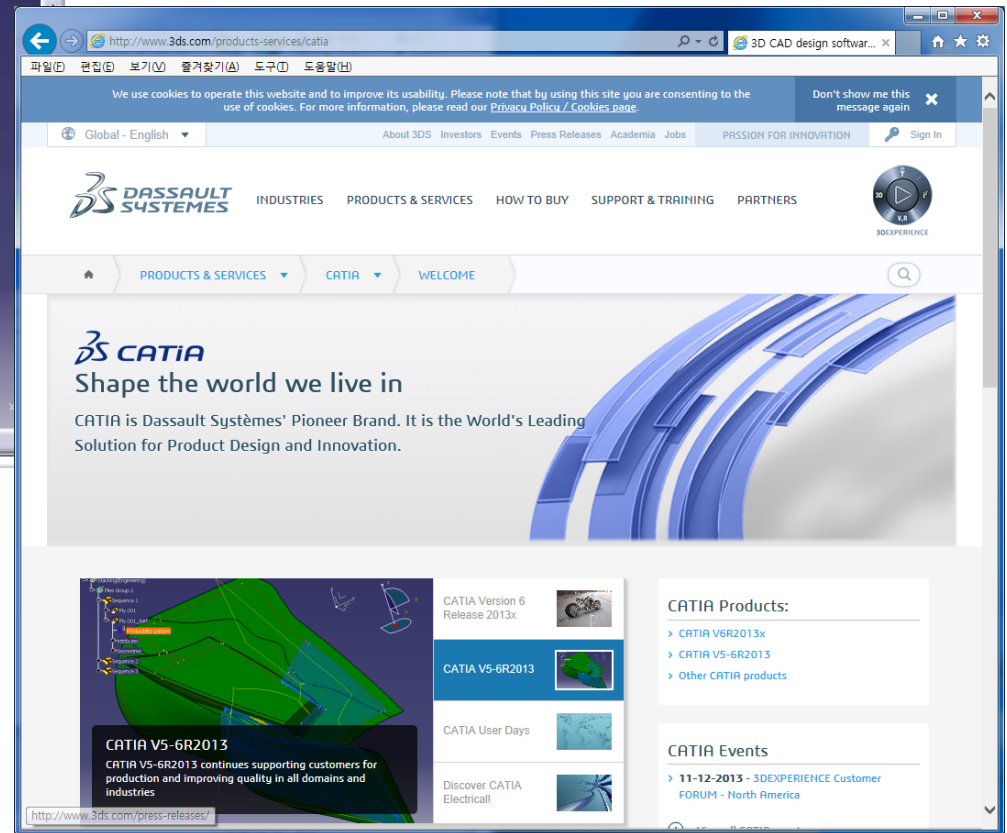
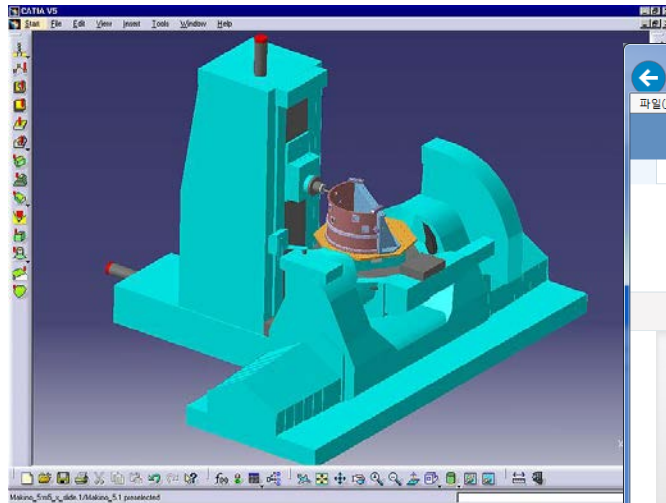
[www.autodesk.com](http://www.autodesk.com)



# I. 3D CAD 의 개요

## 3-3. CATIA

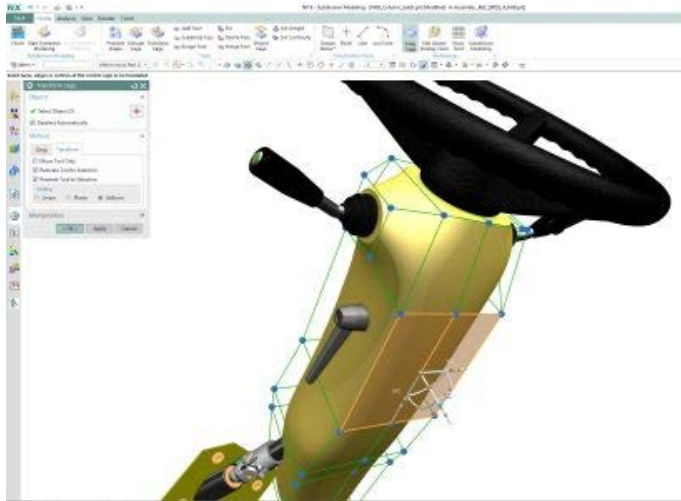
www. catia. com



# I. 3D CAD 의 개요

## 3-4. NX, (UG-NX)

www.plm.automation.siemens.com



SIEMENS

잠재적 기술

Advance 1

Advance 2

Value Gap

Industry Catalysts의 활용

가치

산업 기반의 구현

시간

Siemens PLM Software

전체 메뉴

언락처

홈

제품 및 솔루션을 선택하세요...

- Fibersim
- LMS
- NX
  - Design(CAD)
  - Simulation (CAE)
  - Manufacturing (CAM)
- PLM Components
- Seat Design Environment
- Syncofit
- Tosmcraft

Explore Siemens PLM Software

Siemens PLM Software는 세계 선도의 **제품 라이프사이클 관리(PLM) 소프트웨어** 제공업체입니다. **HD-PLM** 비전을 토대로 개발된 Siemens PLM Software의 PLM 솔루션은 보다 현명한 의사 결정을 내리고 더 나은 제품을 생산할 수 있도록 도와드릴 것입니다.

- About Us
- Facts and Philosophy
- 뉴스룸
- Academic
- 교육
- 지원

주요 기사

- 이벤트 및 프로모션
- PLM Blog

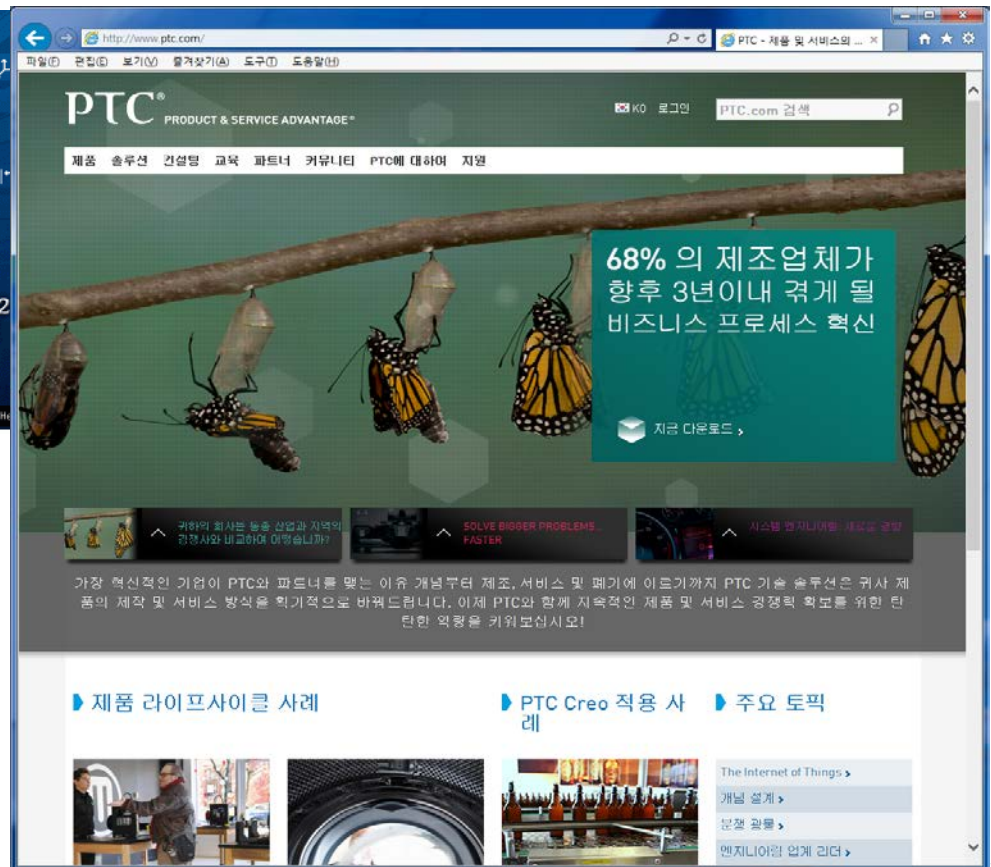
- 지멘스 개인정보처리(취급)방침: 최종 업데이트 2013-09-30
- 미국 행정부 추진 차세대 제조·설계 혁신 허브 '디지털 팩' 최상위 파트너 선정
- 지멘스 PLM 소프트웨어, LMS Imagine.Lab 소프트웨어 최신 버전 출시
- 지멘스 PLM 소프트웨어 항공기 제작사 에어버스에 LMS 솔루션 구축
- 지멘스 PLM 소프트웨어, TESIS Di Mura 이스트...



# I. 3D CAD 의 개요

## 3-5. CREO, PRO ENGINEER (PRO/E)

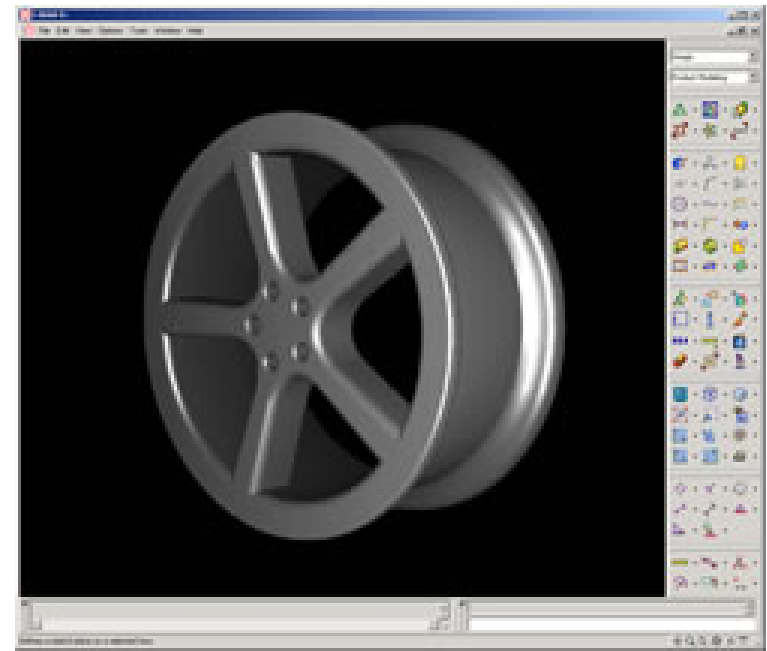
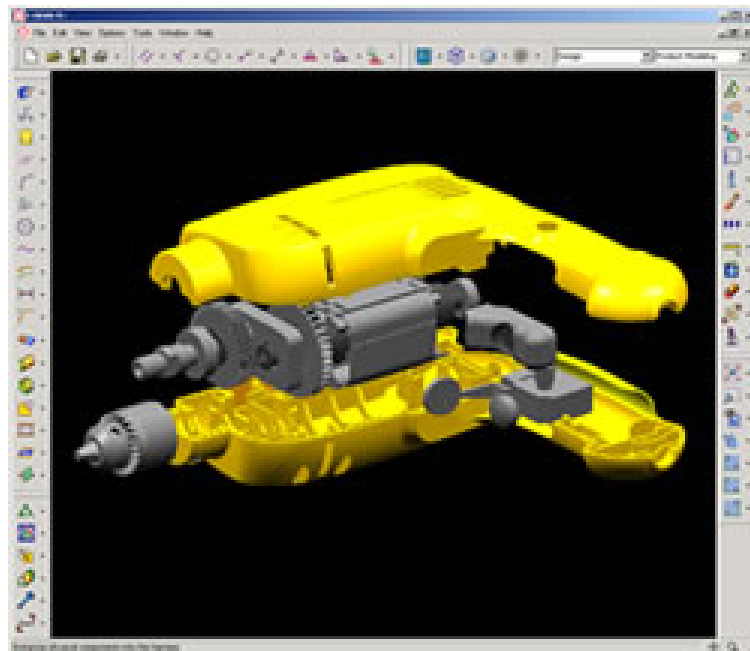
www.ptc.com



# I. 3D CAD 의 개요

## 3-6. I-DEAS

지금은 지멘스 NX site ([www.sdrc.com](http://www.sdrc.com))



## II. CATIA 의 개요

### 1. 3D CAD (CATIA)

- **CATIA**란 Computer-Graphics Aided Three-dimensional Interactive Application의 약자
- **사용분야**는 항공, 건축, 건설, 전자산업, 기구, 기계제조, 금형 업체, 완구류, 장치업, 자동차산업, 소비재산업, 조선업계, 중공업, 의료기기 등에서 사용
- **범용 CAD / CAM / CAE 솔루션으로** 다쏘 시스템즈 (Dassault Systems)가 개발을, IBM이 영업 및 지원을 담당
- **다쏘 시스템즈**는 세계적으로 우주, 항공 산업을 선도하는 다쏘 그룹의 계열사로, 다쏘 에어로스페셜 (Dassault aerospace)사에서 자체 개발한 CAD 소프트웨어를 상용화하면서 탄생된 회사



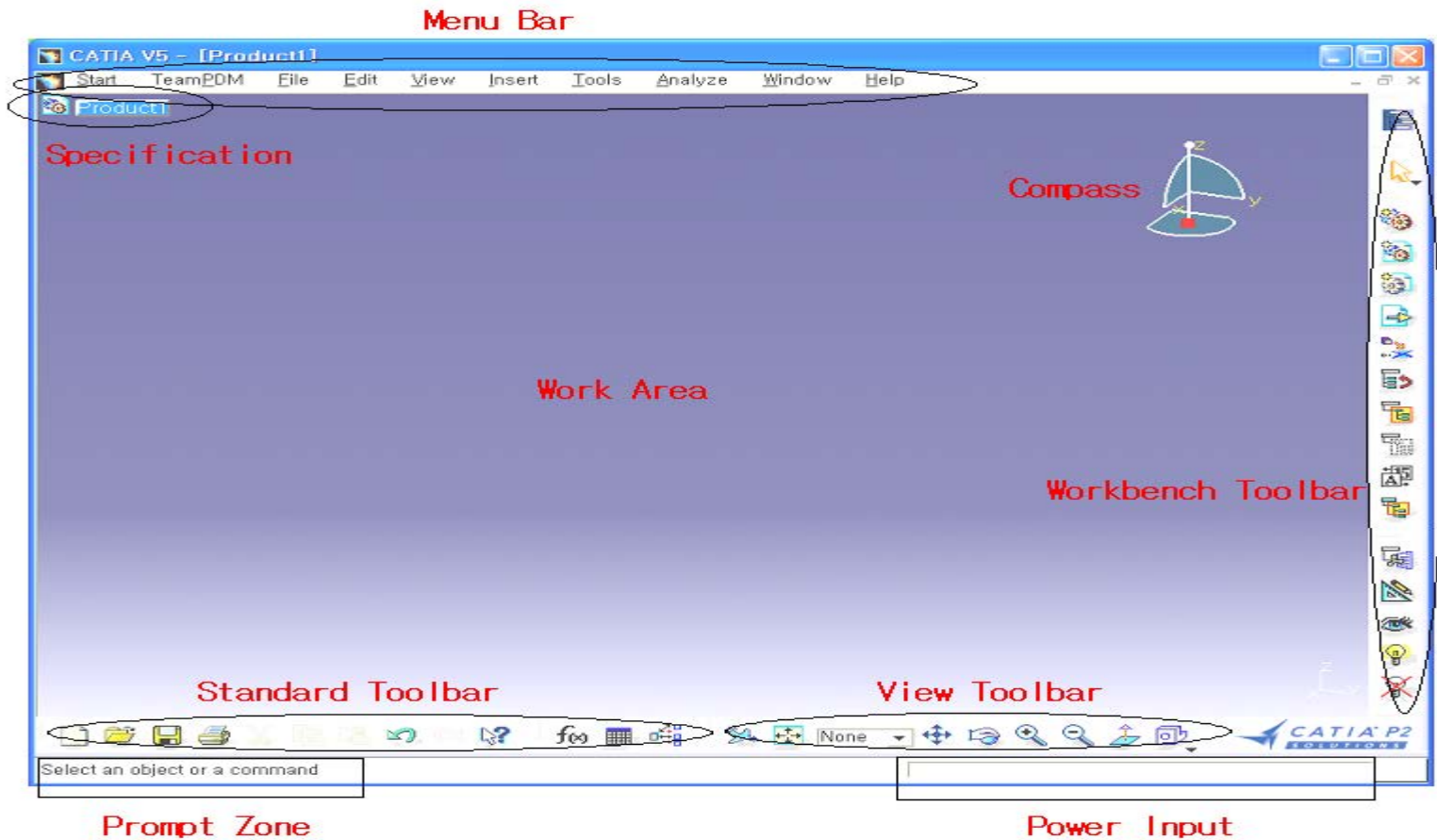
## II. CATIA 의 개요

### 2. CATIA 의 특징 및 구성

- 1) 서페이스와 솔리드가 하나의 통합된 시스템에서 운영되는 하이브리드(Hybrid) 모델링.  
2) 일관되고 직관적인 사용자 환경을 제공하여 학습 및 사용이 용이  
3) OpenGL 을 사용한 고성능의 그래픽 환경을 제공함으로써 상호 대화적으로 제품을 설계할 수 있으며, 실시간으로 형상의 품질이나 설계의 변경
- **Version 5** 는 크게 세 부분의 Platform으로 나누어지고 각 Plat-form마다 사용하는 기능이 다르며, 옵션 창에서 General 을 선택하고 General 의 Users Interface Style 의 CATIA-P1, CATIA-P2 , CATIA-P3 를 선택
- *Mechanical Design   Shape Design and Styling*
- *Analysis Product Synthesis   Plant Design*
- *NC Manufacturing Infrastructure*
- *Equipment and Systems Engineering* 등의 Product

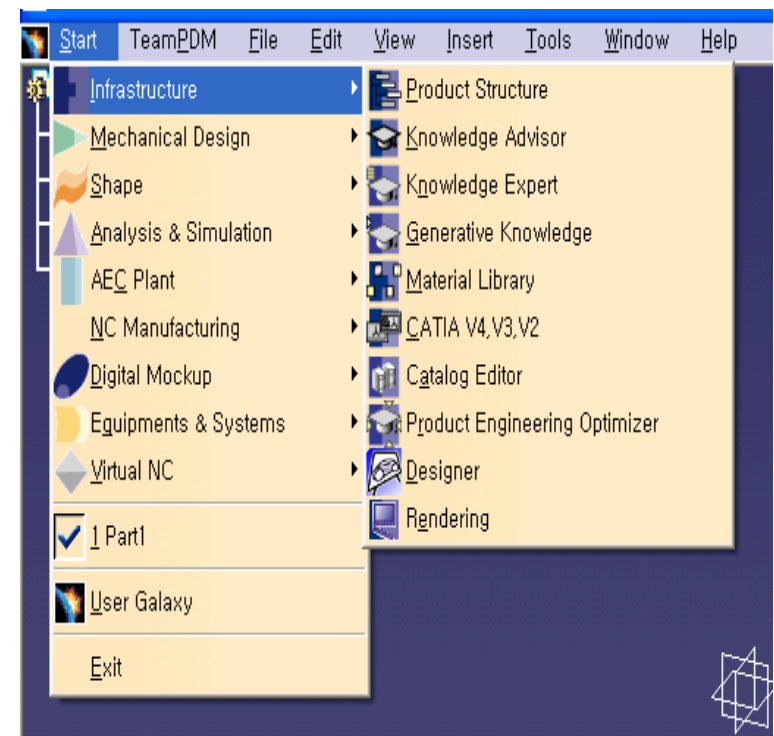
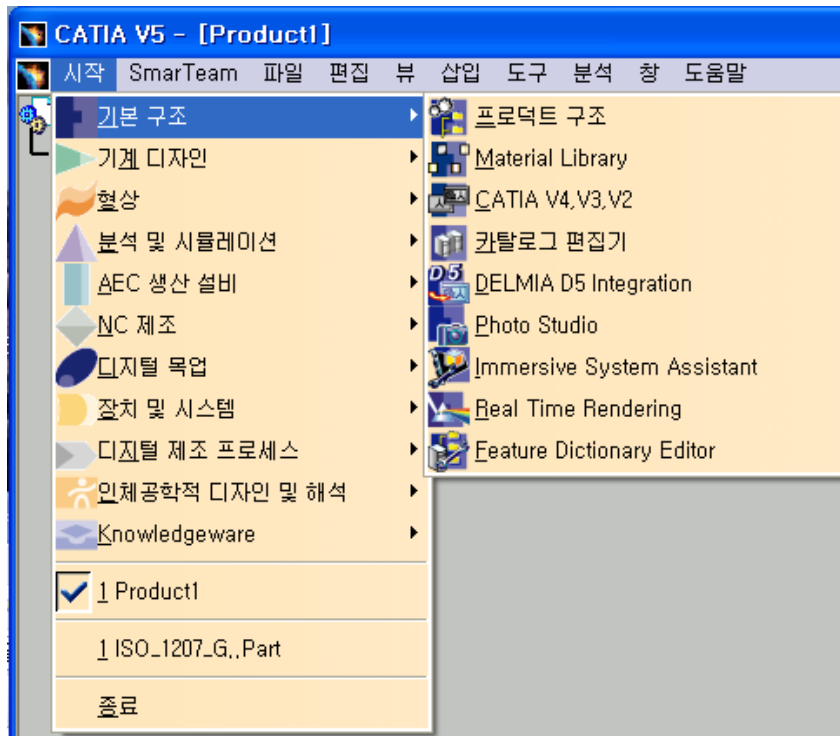
# III. CATIA 의 구성 및 기초

## 1. CATIA 의 화면구성



# III. CATIA 의 구성 및 기초

## 2. 메뉴 바의 내용



# III. CATIA 의 구성 및 기초

## 2-1. 시작 (Start) (1)

### 기본 구조

- 프로젝트 구조
- Material Library
- CATIA V4, V3, V2
- 카탈로그 편집기
- DELMIA D5 Integration
- Photo Studio
- Immersive System Assistant
- Real Time Rendering
- Feature Dictionary Editor

### 기계 디자인

- Part Design
- 어셈블리 디자인
- Sketcher
- Product Functional Tolerancing & Annotation
- Weld Design
- Mold Tooling Design
- Structure Design
- Drafting
- Core & Cavity Design
- Healing Assistant
- Functional Molded Part
- Sheet Metal Design
- Aerospace Sheet Metal Design
- Sheet Metal Production
- Composites Design
- Wireframe and Surface Design
- Generative Sheetmetal Design
- Functional Tolerancing & Annotation

### 현상

- FreeStyle
- Automotive BiW Fastening
- Sketch Tracer
- Digitized Shape Editor
- Generative Shape Design
- Quick Surface Reconstruction
- Automotive Class A
- Shape Sculptor

# III. CATIA 의 구성 및 기초

## 2-1. 시작 (Start) (2)

### 분석 및 시뮬레이션

- Tolerance Analysis of Deformable Assembly
- Advanced Meshing Tools
- Generative Structural Analysis

### AEC 생산 설비

- Plant Layout

### NC 제조

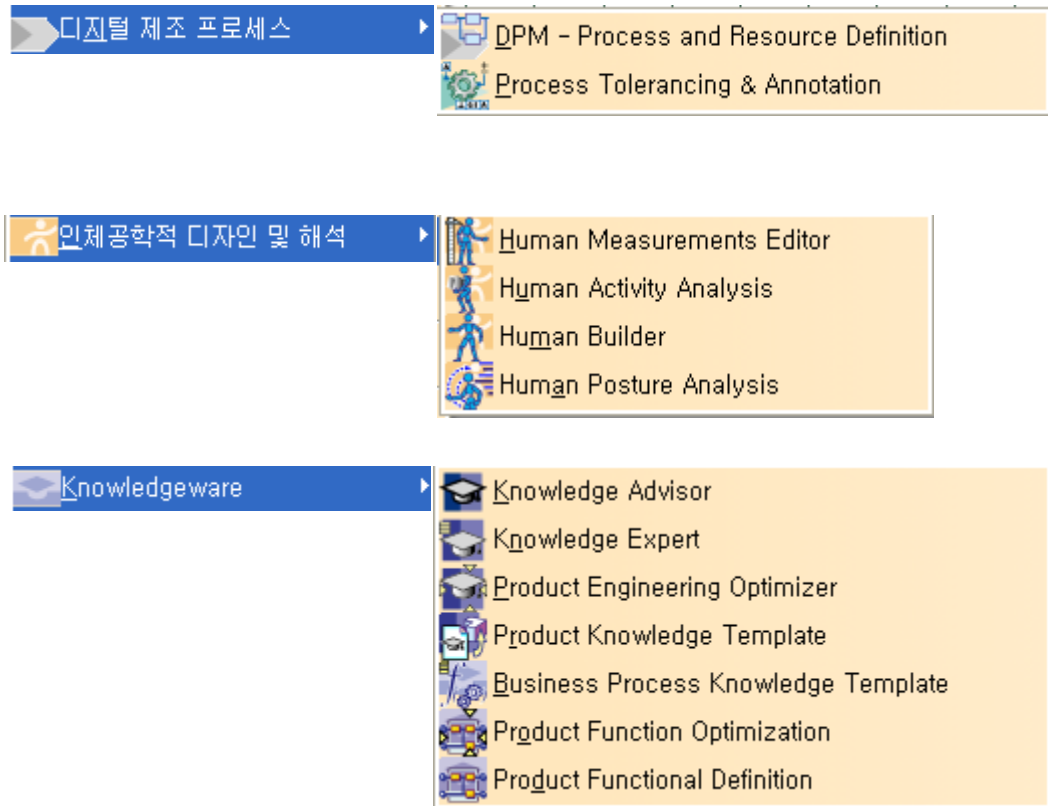
- Lathe Machining
- Prismatic Machining
- Surface Machining
- Advanced Machining
- NC Manufacturing Review
- STL Rapid Prototyping

### 디지털 목업

- DMU Navigator
- DMU Space Analysis
- DMU Kinematics
- DMU Fitting
- DMU 2D 뷰어
- DMU Optimizer
- DMU Tolerancing Review

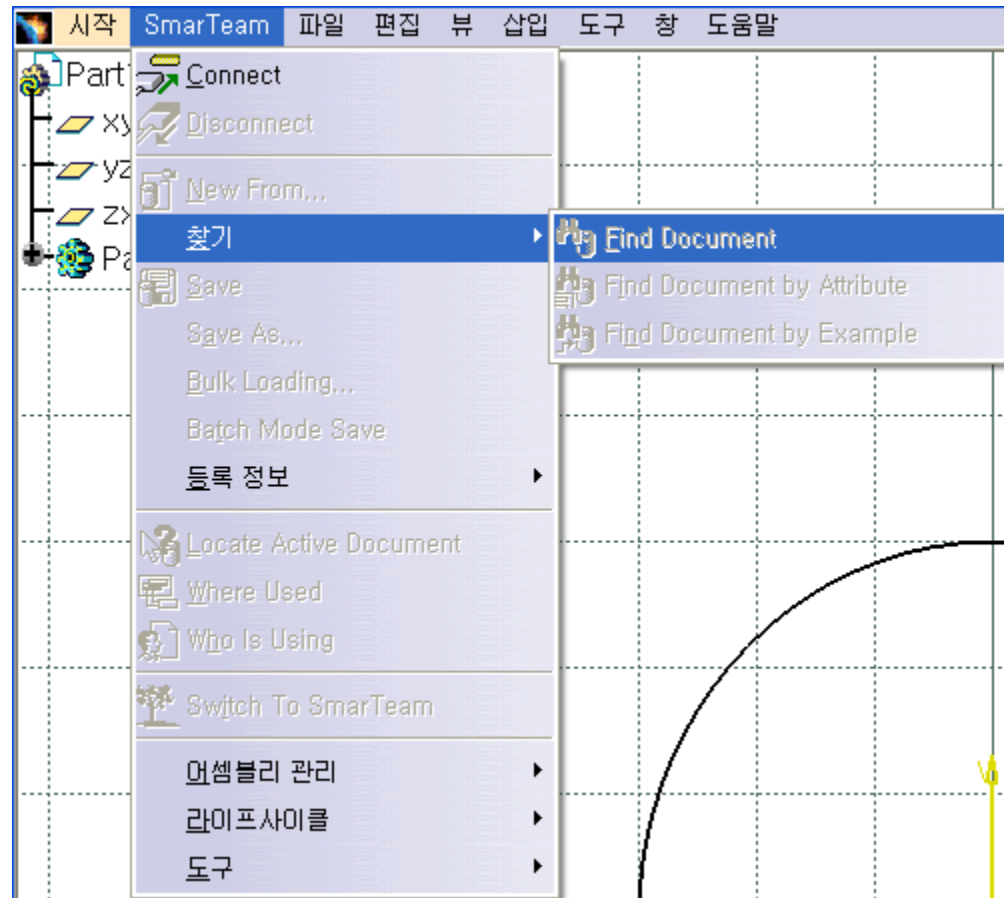
# III. CATIA 의 구성 및 기초

## 2-1. 시작 (Start) (3)



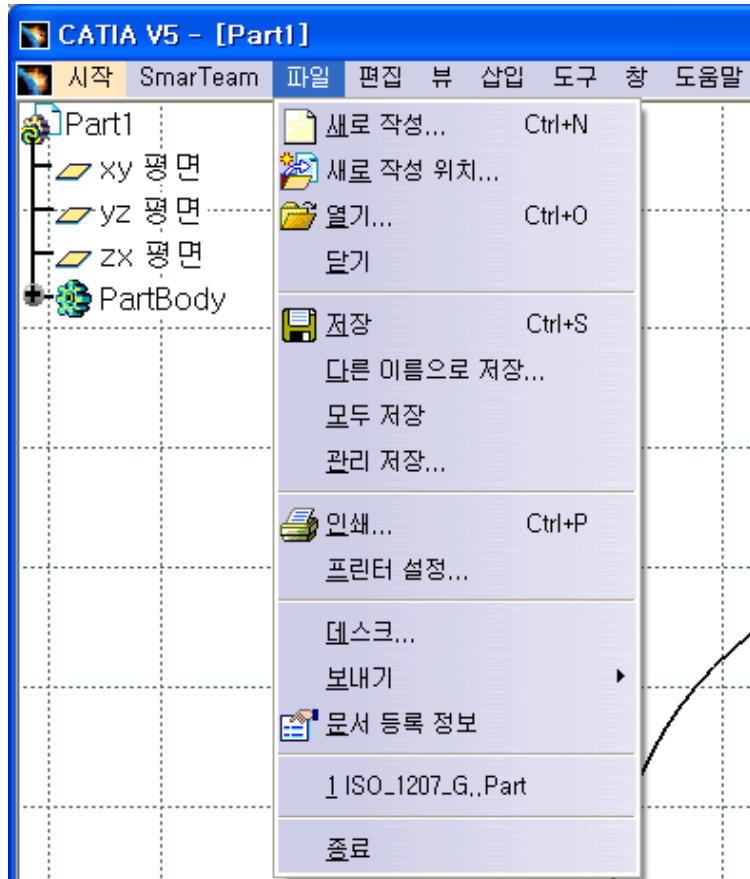
# III. CATIA 의 구성 및 기초

## 2-2. 제품정보관리 (SmarTEAM)

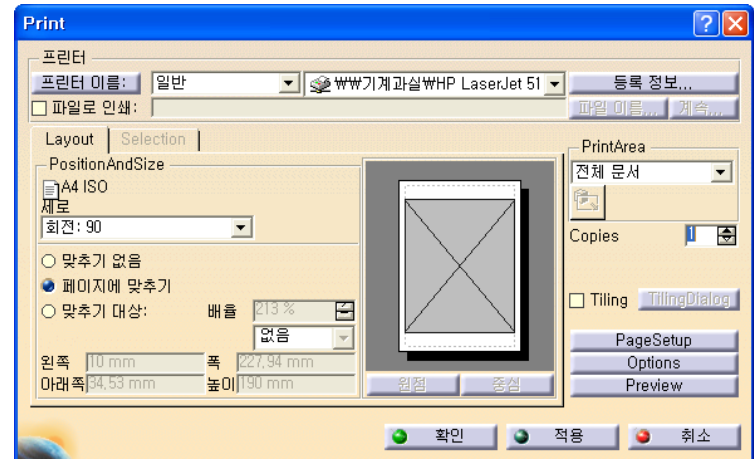
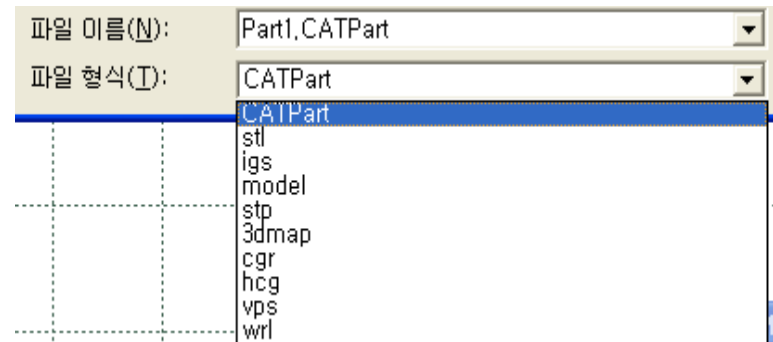


# III. CATIA 의 구성 및 기초

## 2-3. 파일 (File)



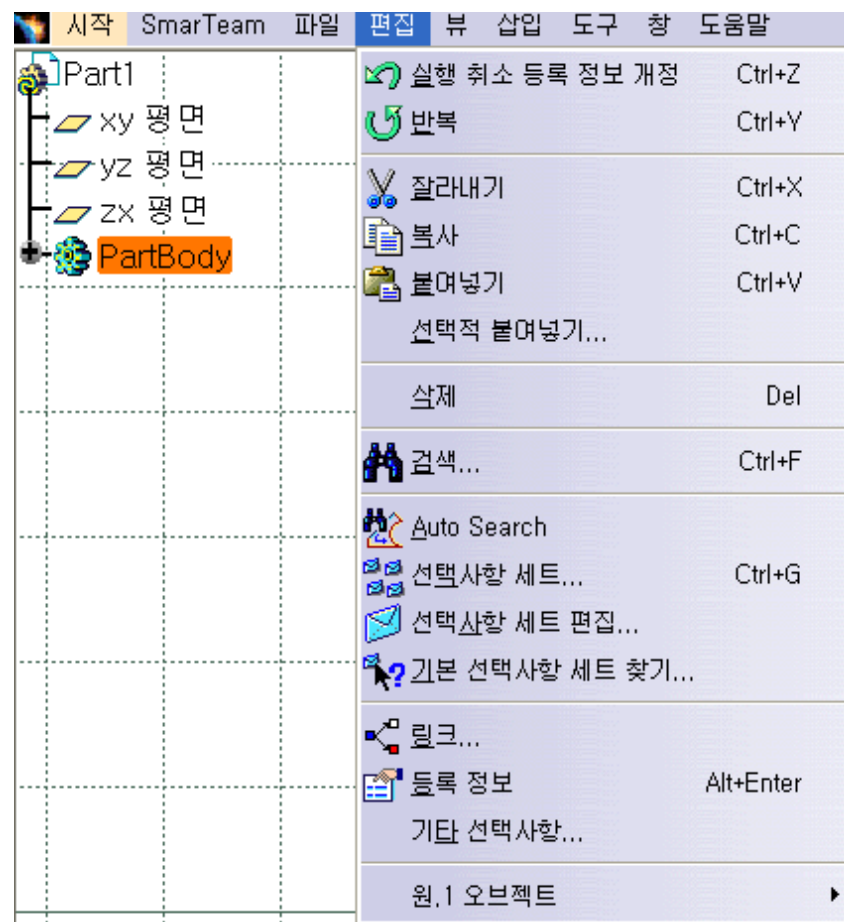
### 다른 이름으로 저장





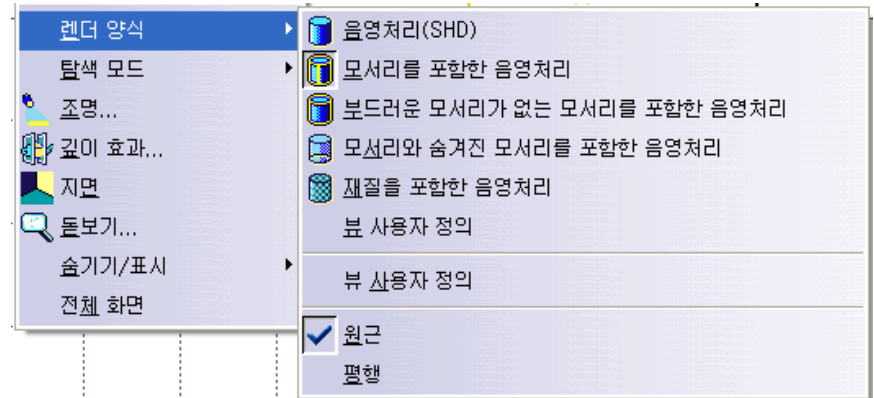
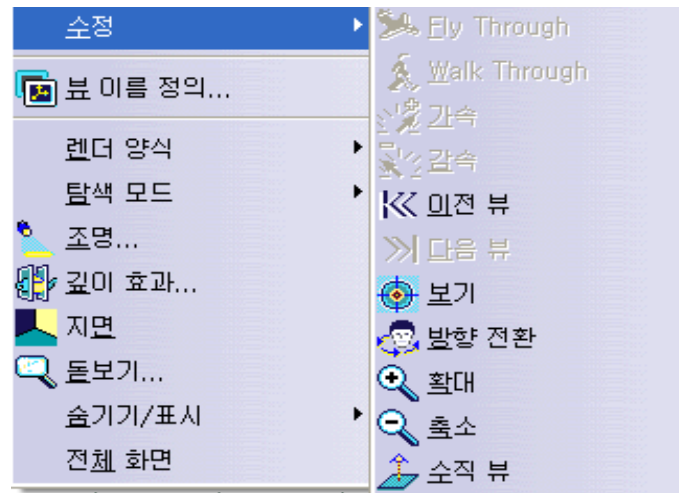
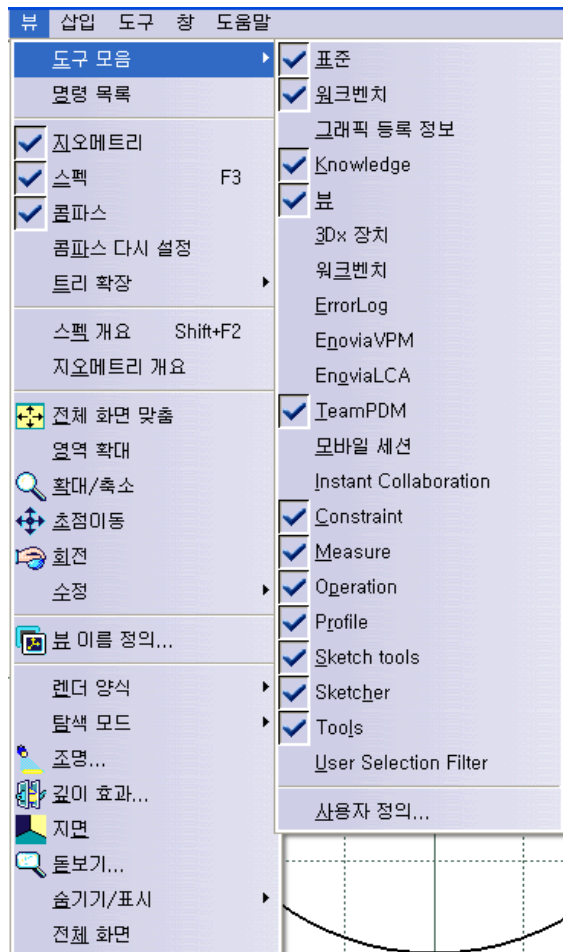
# III. CATIA 의 구성 및 기초

## 2-4. 편집 (Edit)



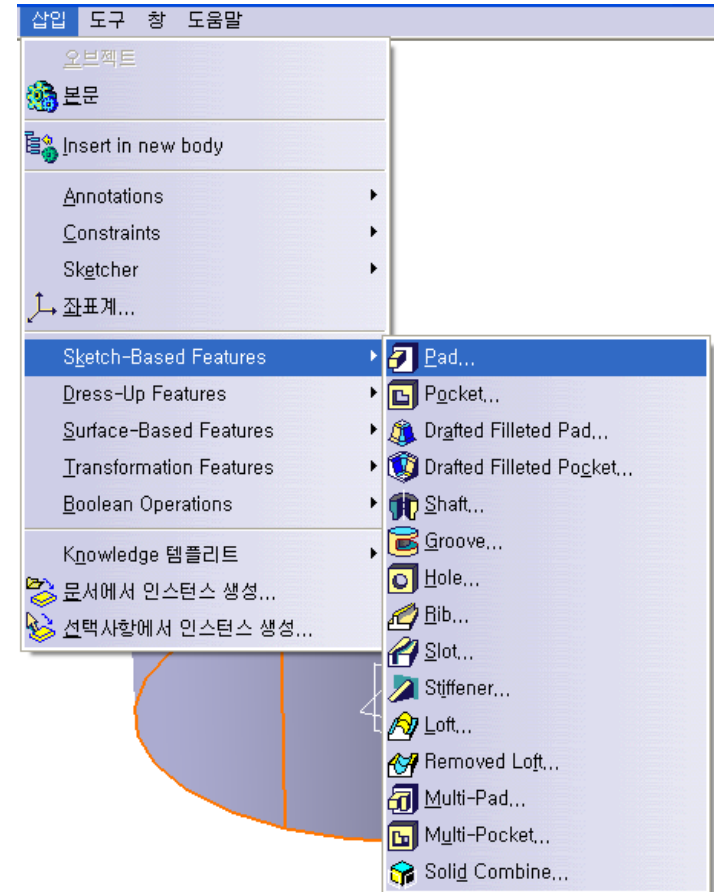
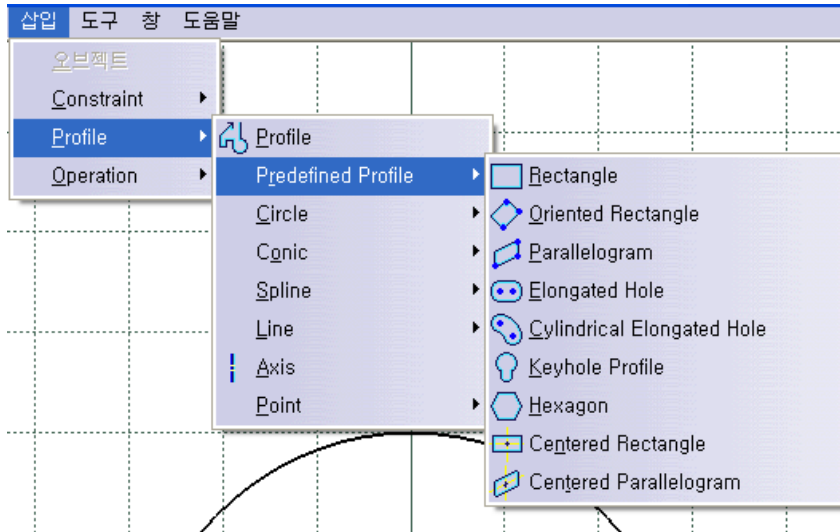
# III. CATIA 의 구성 및 기초

## 2-5. 뷰 (View)



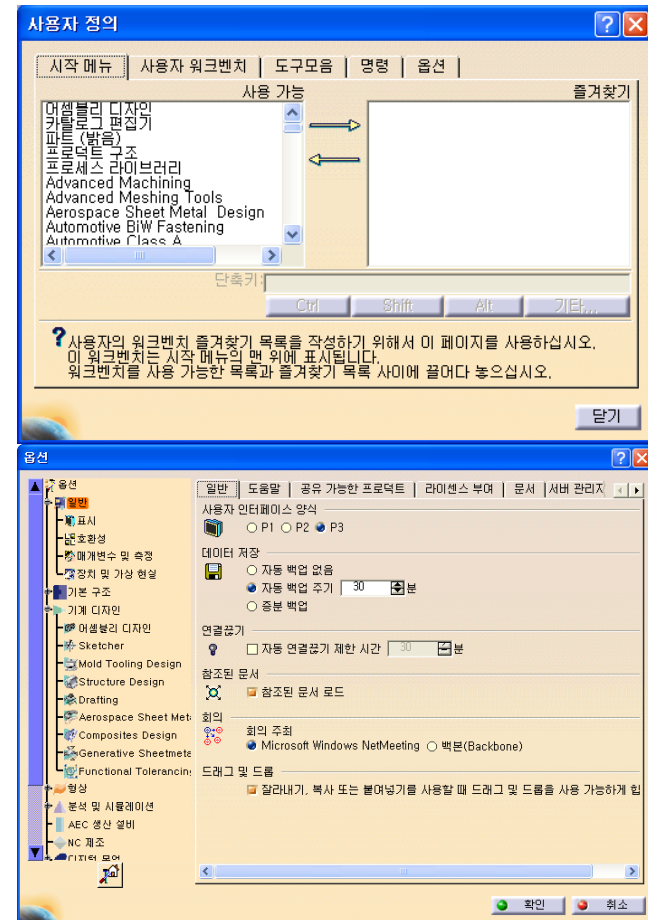
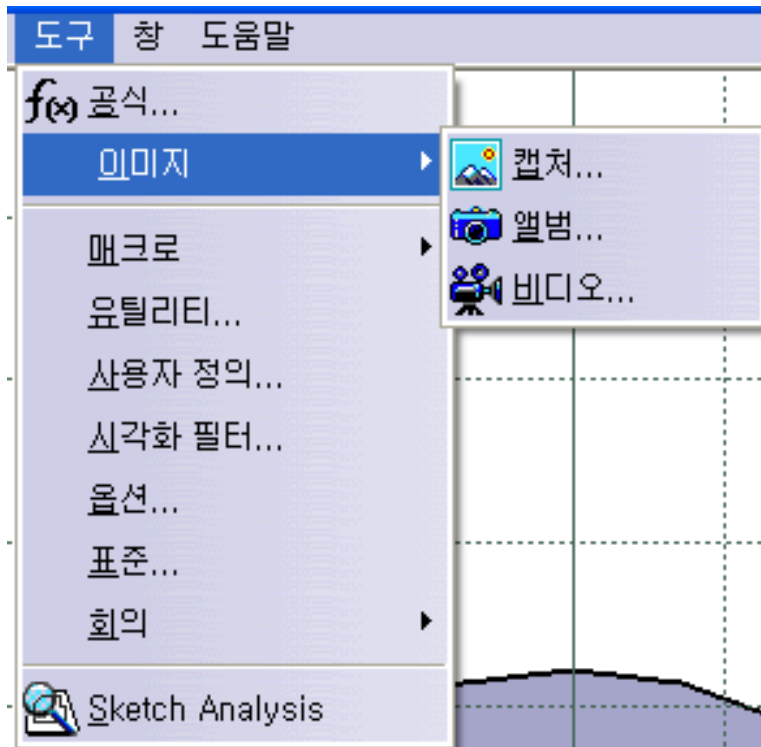
# III. CATIA 의 구성 및 기초

## 2-6. 삽입 (Insert)



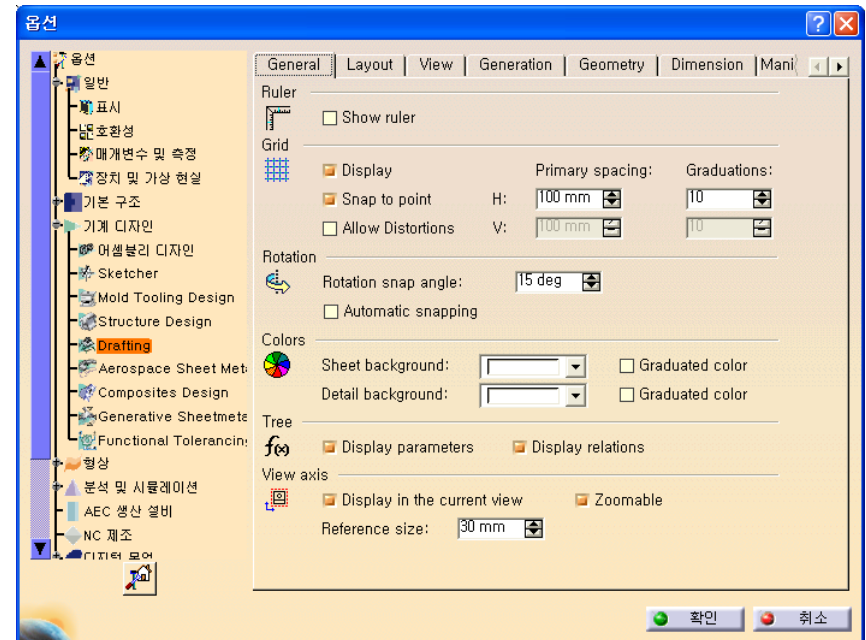
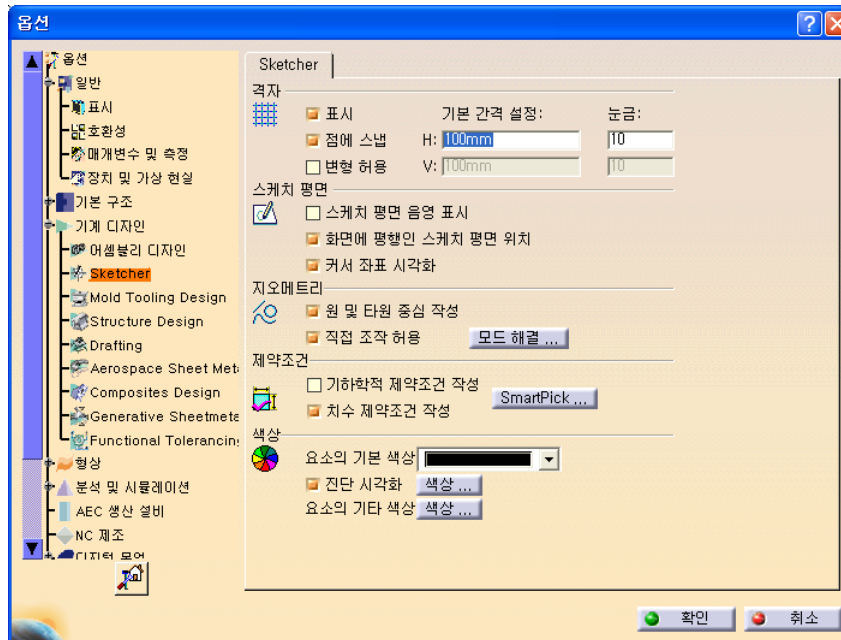
# III. CATIA 의 구성 및 기초

## 2-7. 도구 (Tools)



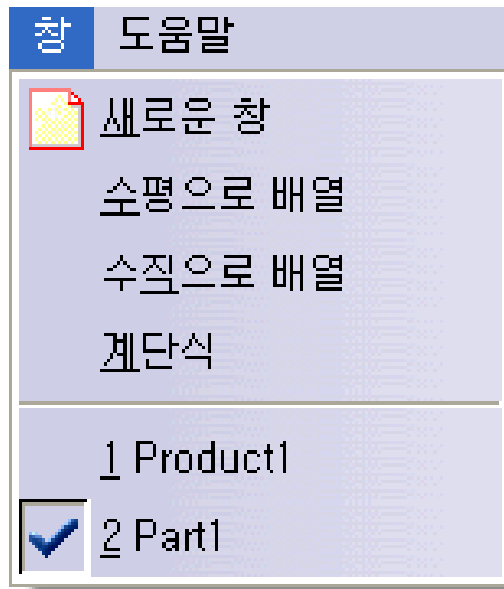
# III. CATIA 의 구성 및 기초

## 2-7. 도구 (Tools) : 옵션 기계디자인의 Sketcher 와 Drafting




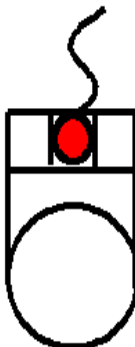

# III. CATIA 의 구성 및 기초

## 2-8. 창 (Windows) 과 도움말 (Help)



# III. CATIA 의 구성 및 기초

## 3. CATIA 의 마우스 사용법











마우스 1 번	마우스 2 번	마우스 3 번
		
대상물과 Icon 선택 Drag & Drop 기능	Pan 기능 과 보조적 기능	대상물에 대한 팝업 메뉴 표시

기 능	사 용 방 법
Pan	마우스 2번 클릭 → Drag
Rotate	마우스 2번 클릭 → 마우스 1번 동 시 클릭 → Drag
Zoom In Zoom Out	마우스 2번 클릭 → 마우스 1번 클 릭 후 뽕 → Drag



# III. CATIA 의 구성 및 기초

## 4. CATIA 의 저장 파일

<p><u>.CATPart</u></p>	<ul style="list-style-type: none"> <li> Part Design</li> <li> Sketcher</li> <li> Sheet Metal Design</li> <li> Sheet Metal Production</li> <li> Wireframe and Surface Design</li> <li> Functional Tolerancing &amp; Annotation</li> </ul>
<p><u>.CATProduct</u></p>	<ul style="list-style-type: none"> <li> Assembly Design</li> <li> Mold Tooling Design</li> <li> Structure Design</li> </ul>
<p><u>.CATDrawing</u></p>	<ul style="list-style-type: none"> <li> Drafting</li> </ul>



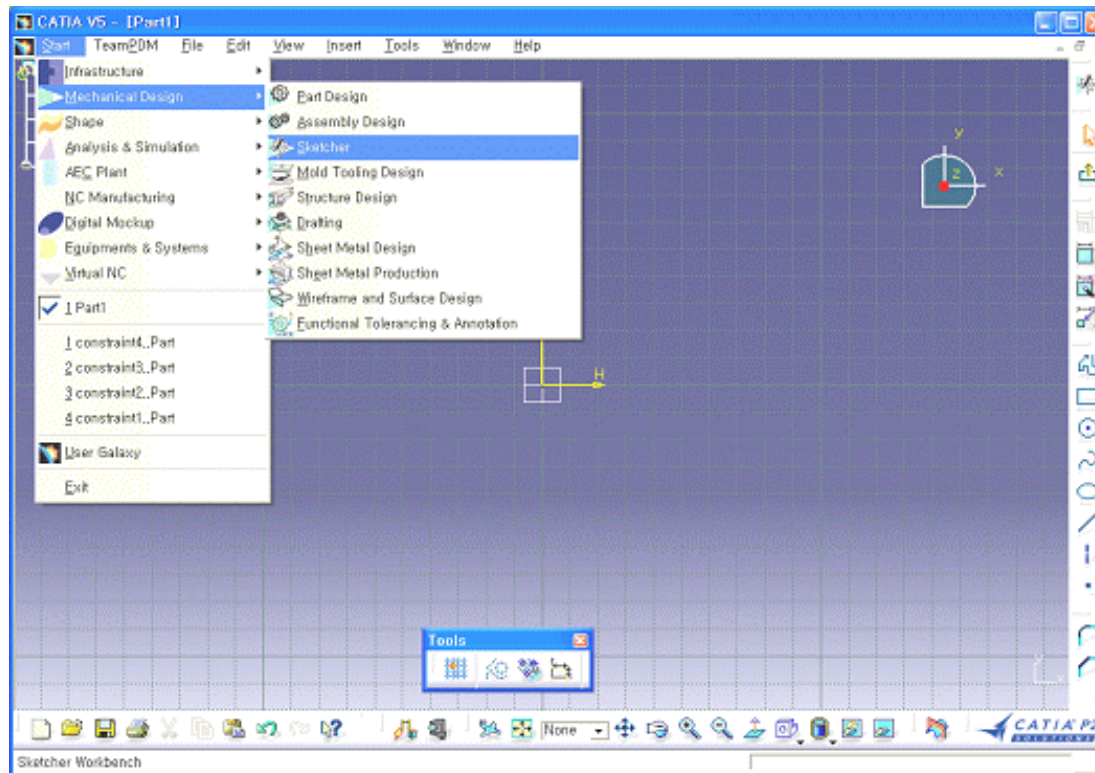
## IV. SKETCHER

### 1. Sketcher 의 소개



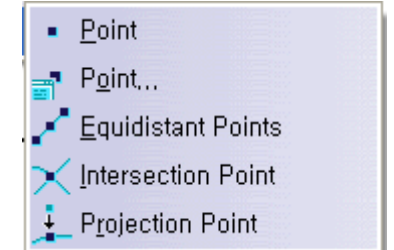
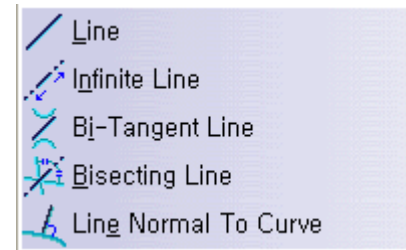
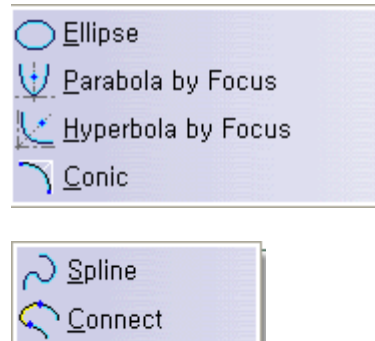
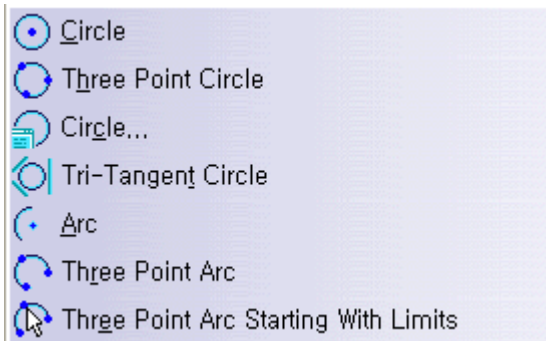
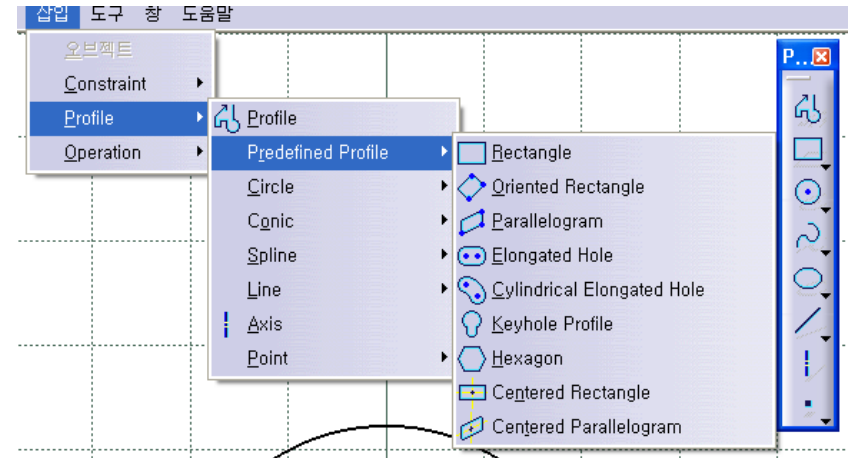
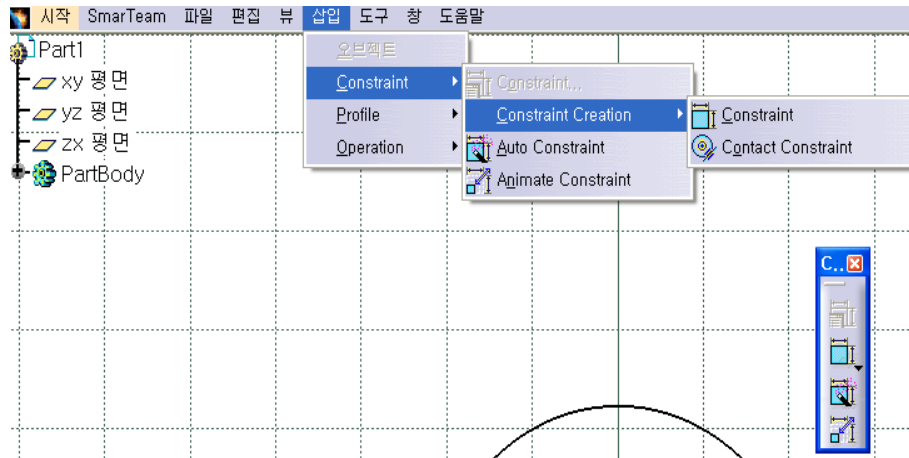
Sketcher

Sketcher 의 시작이나 이동



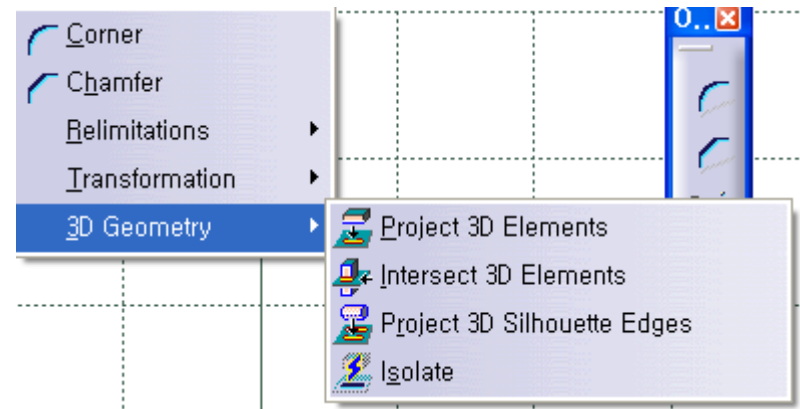
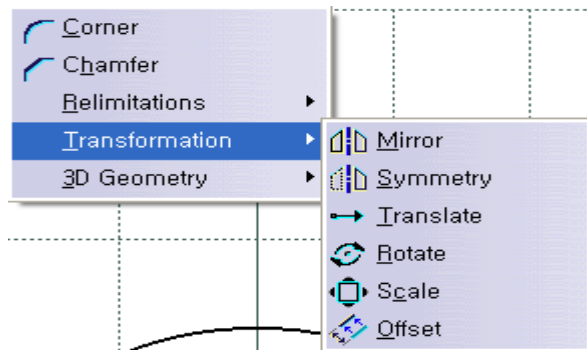
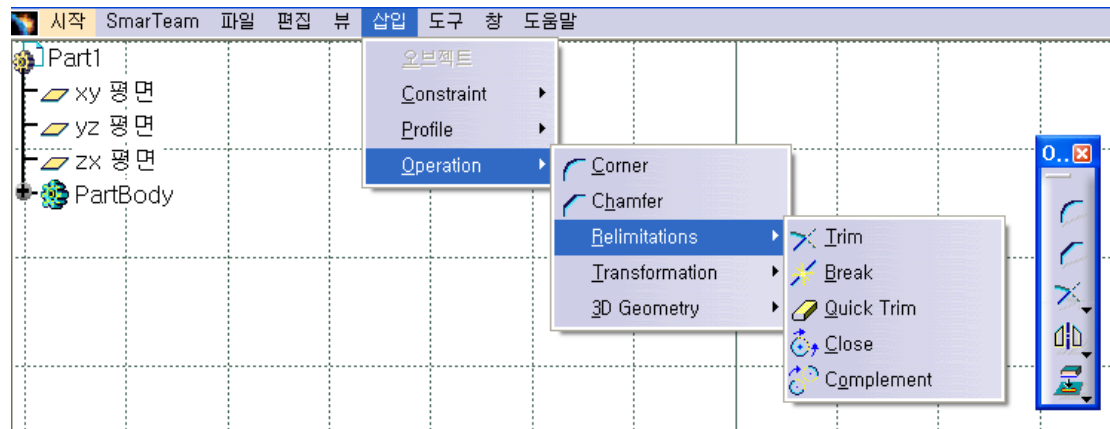
# IV. SKETCHER

## 2. Sketcher 의 Insert 명령과 Icon (1)



## IV. SKETCHER

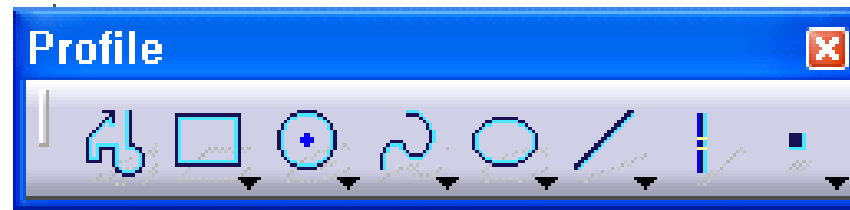
### 2. Sketcher 의 Insert 명령과 Icon (2)



## IV. SKETCHER


### 3. Sketching Simple Profile, Profile

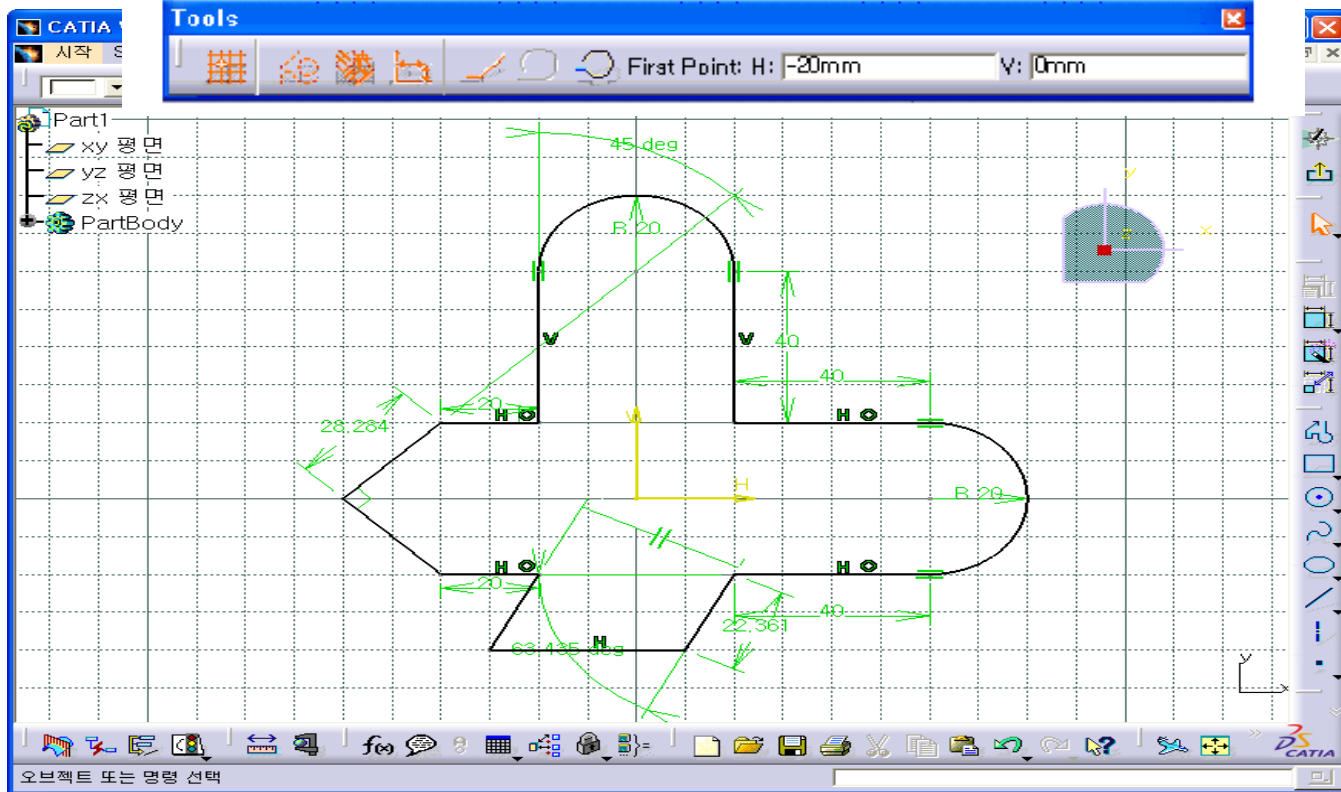
Sketcher 에서 Profile을 생성하는 가장 기본적인 툴 바다.



# IV. SKETCHER

## 3-1. Profile

	<b>PROFILE</b>	Arc, Line, Circle을 혼합하여 생성
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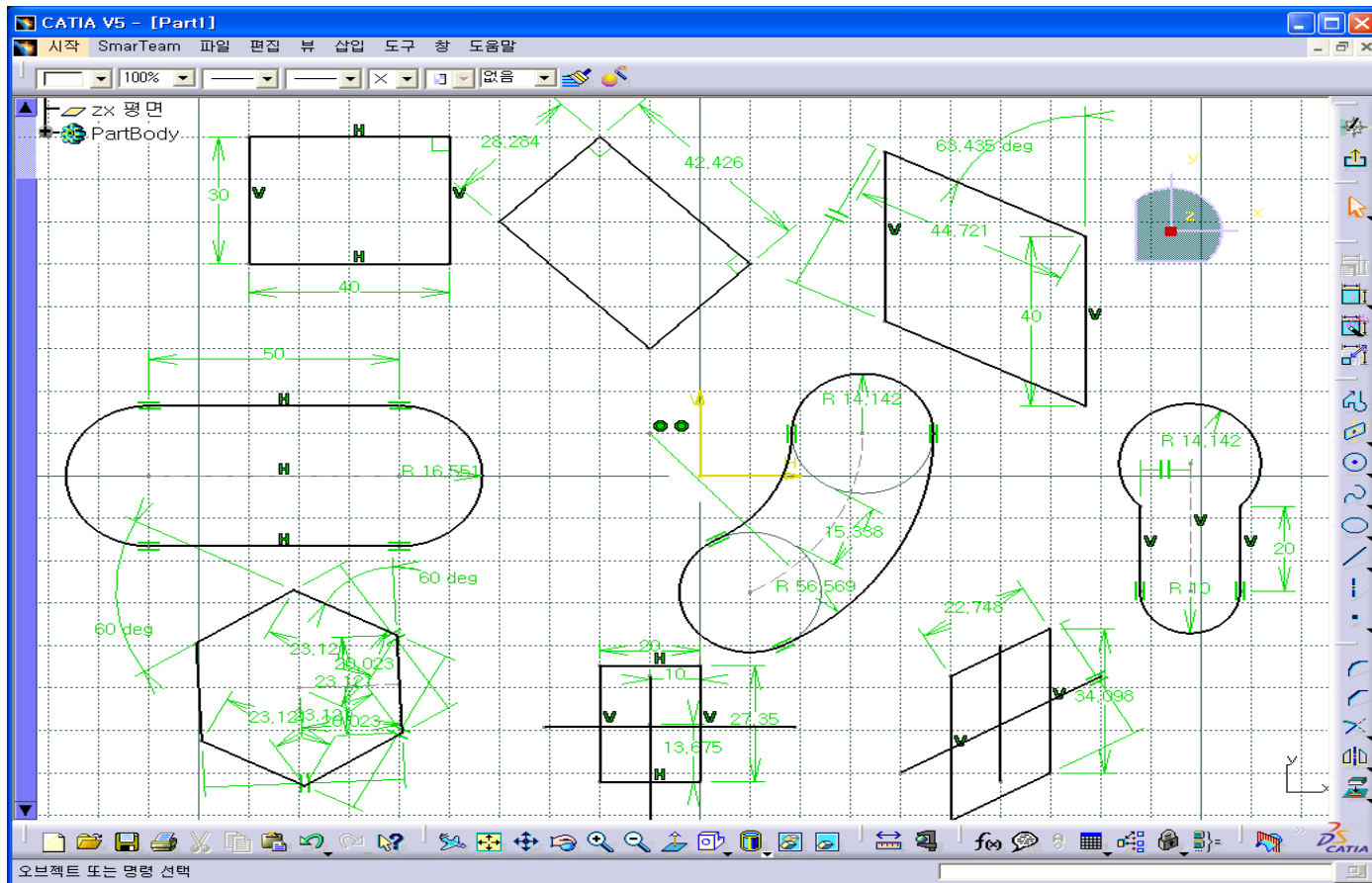
## IV. SKETCHER

### 3-2. Rectangle (1)



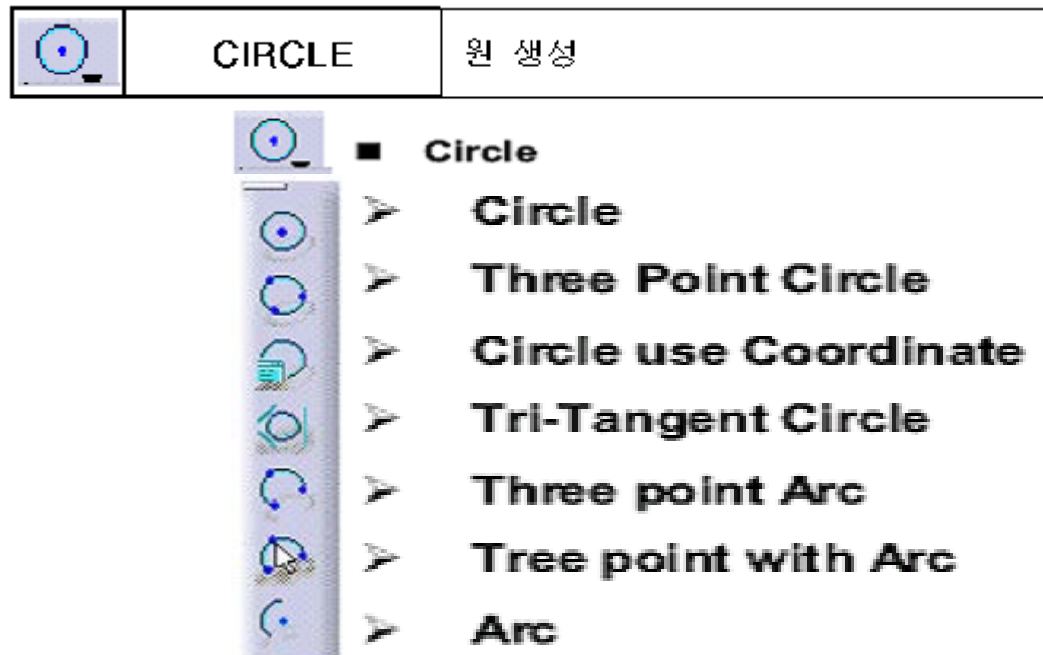
## IV. SKETCHER

### 3-2. Rectangle (2)



## IV. SKETCHER

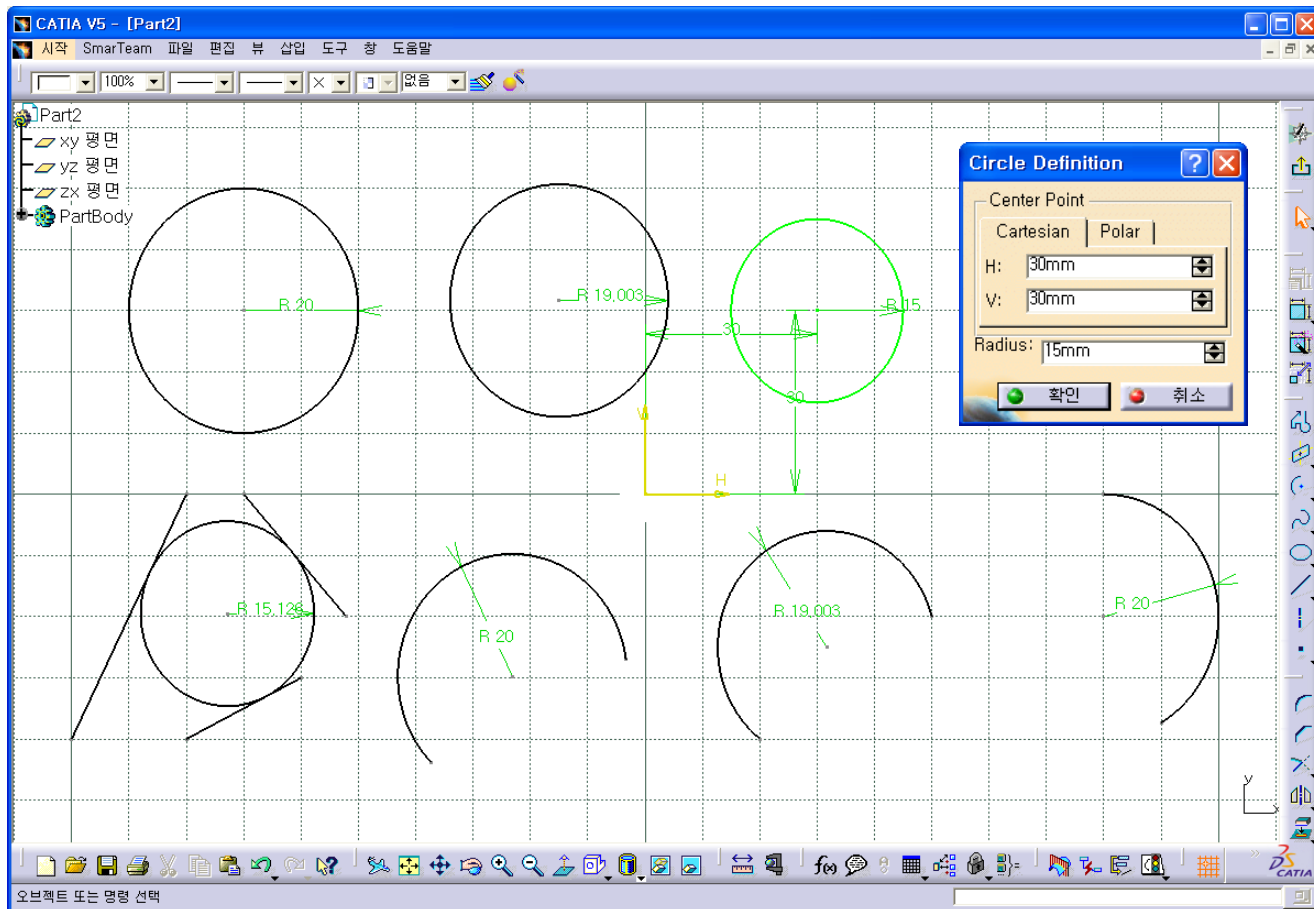
### 3-3. Circle (1)





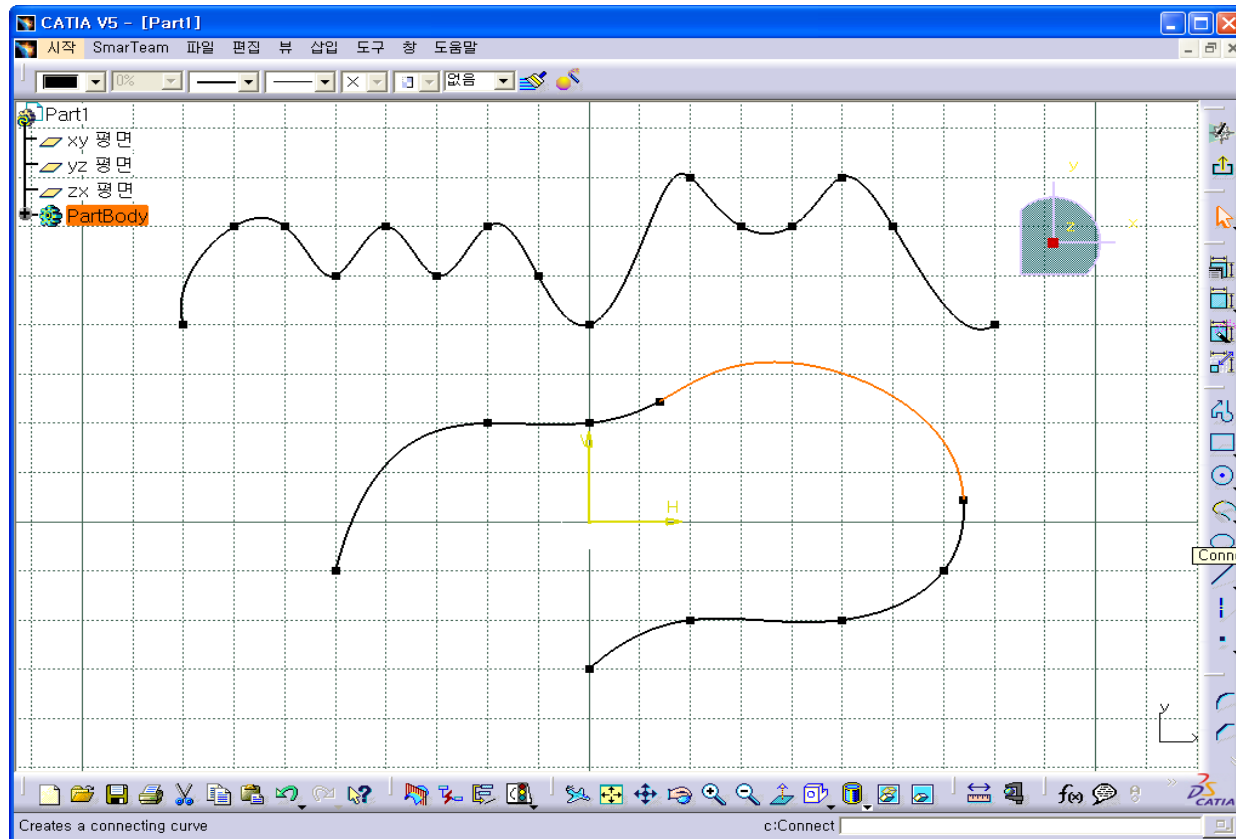
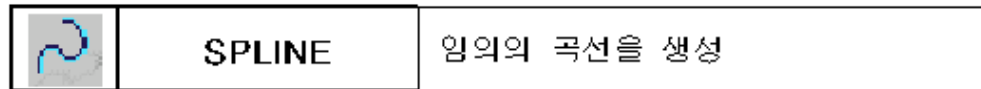
# IV. SKETCHER

## 3-3. Circle (2)




## IV. SKETCHER

### 3-4. Spline







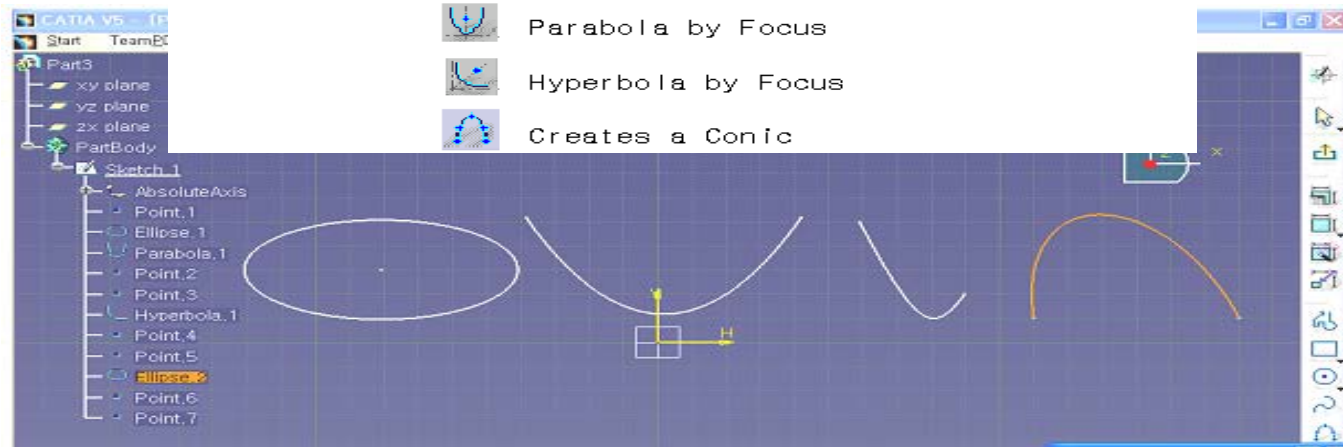
# IV. SKETCHER

## 3-5. Ellipse

	ELLIPSE	타원을 생성
---	---------	--------

 Ellipse  
 Parabola by Focus  
 Hyperbola by Focus  
 Creates a Conic



**Ellipse Definition**

Center Point  
Cartesian | Polar |

H: -80.927mm  
V: 30.117mm

Major radius: 40mm  
Minor radius: 20.656mm  
Angle: 0deg

☐ Construction element

OK Cancel

**Parabola Definition**

Focus Point  
Cartesian | Polar |

H: 1.895mm  
V: 21.829mm

Apex Point  
Cartesian | Polar |

H: 11.685mm  
V: 11.712mm

☐ Construction element

OK Cancel

**Hyperbola Definition**

Focus Point  
Cartesian | Polar |

H: 80.647mm  
V: 11.652mm

Center Point  
Cartesian | Polar |

H: 30.647mm  
V: -20.059mm

Excentricity: 1.057054232

☐ Construction element

OK Cancel

**Ellipse Definition**

Center Point  
Cartesian | Polar |

H: 143.085mm  
V: -2.19mm

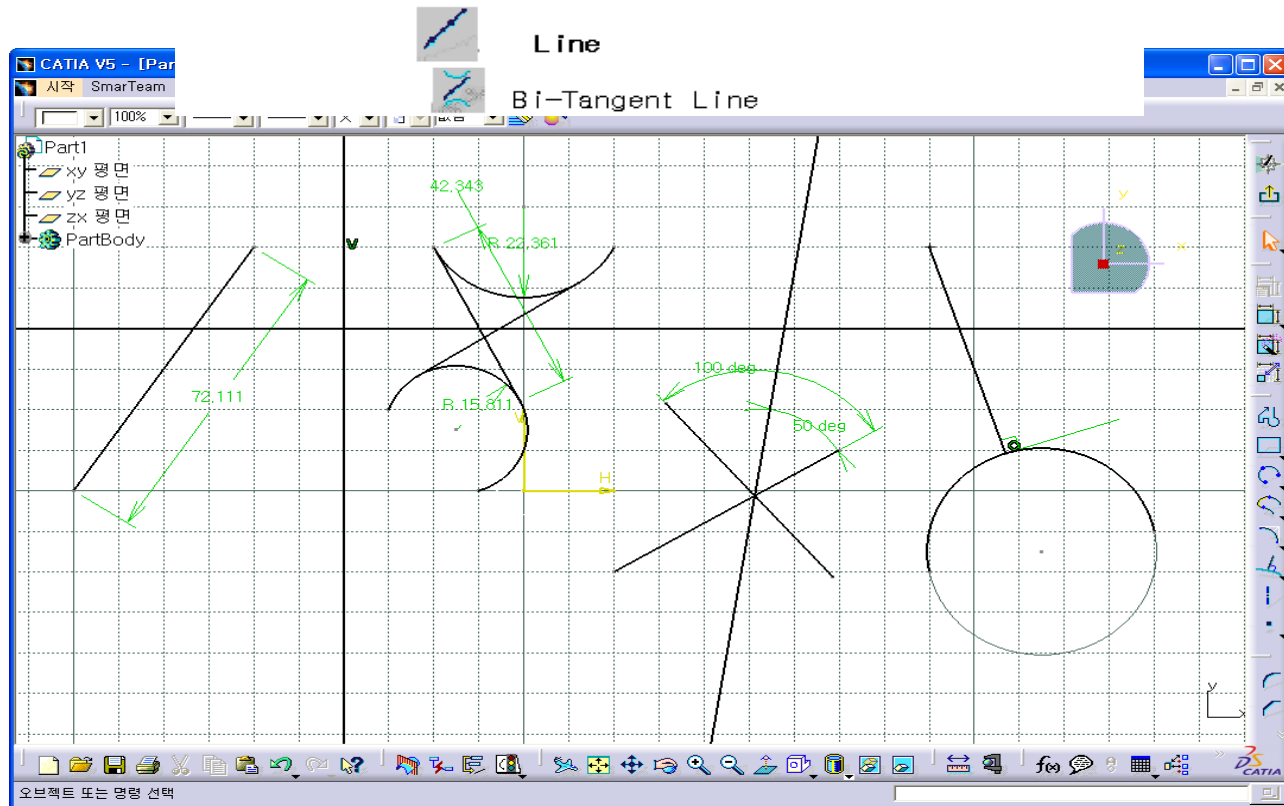
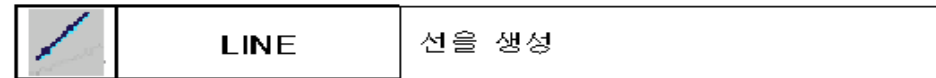
Major radius: 57.506mm  
Minor radius: 29.506mm  
Angle: -70.772deg

☐ Construction element

OK Cancel

# IV. SKETCHER

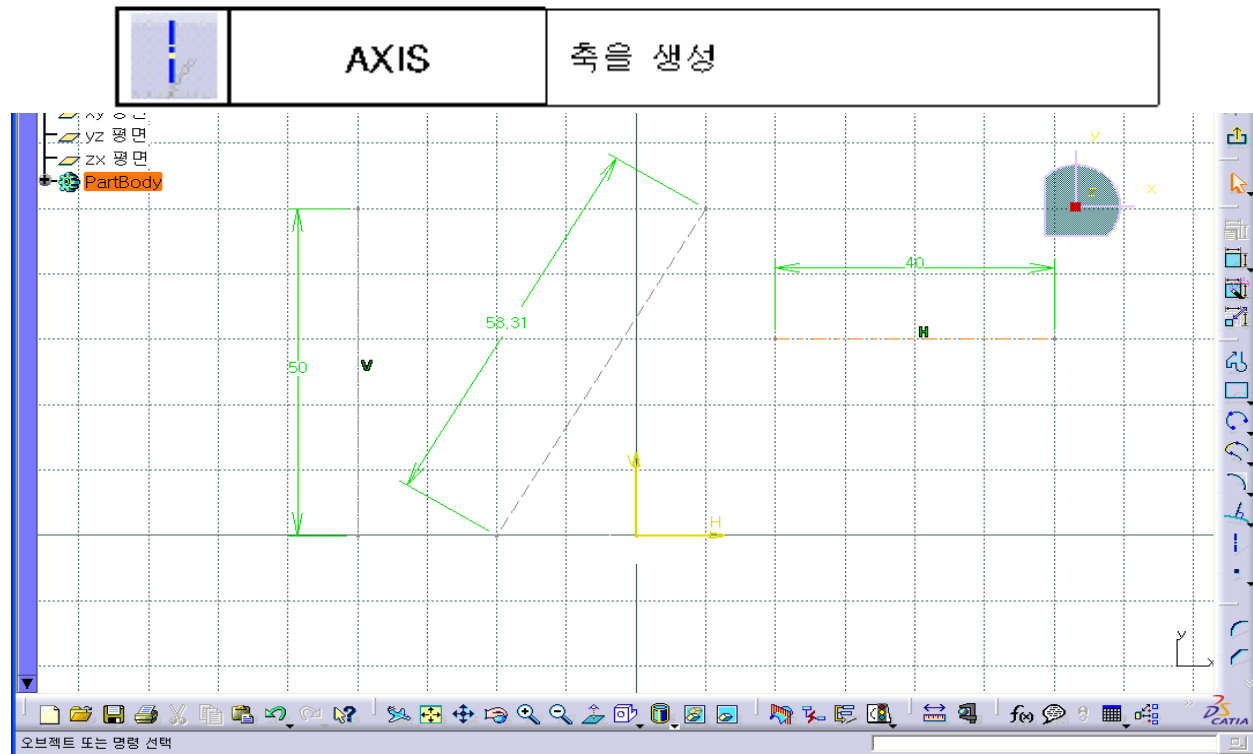
## 3-6. Line



# IV. SKETCHER

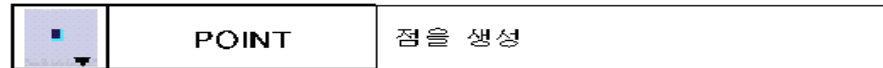
## 3-7. Axis

축을 생성해준다. 보조 Profile 로 일점쇄선으로 나타나며 회전축이 필요한 솔리드나 서페이스 등을 만들 때 유용하게 쓰인다. 또한, 3차원 작업창에서는 나타나지 않는다.

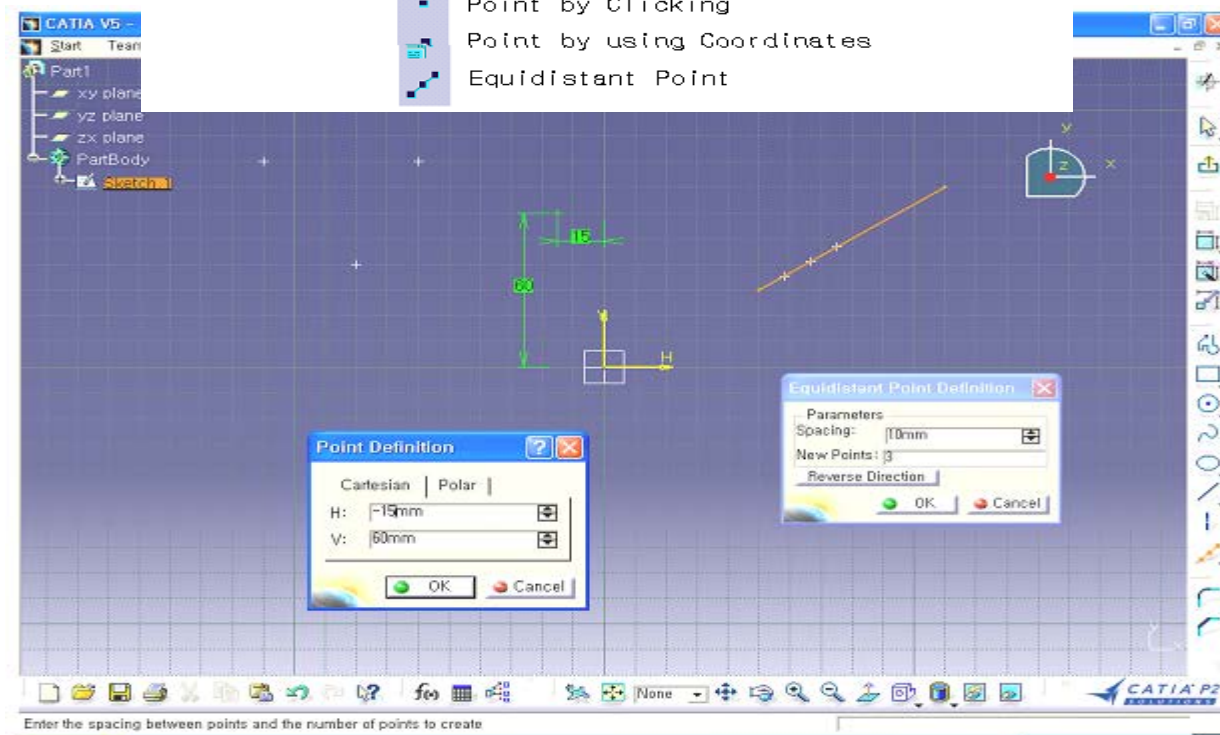


# IV. SKETCHER

## 3-8. Point



- Point
- Point by Clicking
- Point by using Coordinates
- Equidistant Point



## IV. SKETCHER

### 4. Setting Constraints

Sketcher 에서 생성한 Profile 에 치수나 구속 조건 ( dimensional and geometric constraints ) 을 생성하는 기본적인 툴 바다.



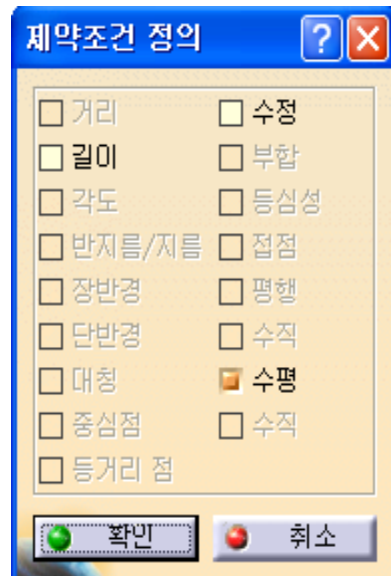
<b>녹 색</b>	<b>정상적인 치수</b>	
<b>짙은 녹색</b>	<b>고정된 치수</b>	<b>치수의 삭제</b>
<b>보라색</b>	<b>필요치 않은 치수 생성</b>	<b>보라색의 치수 중 하나를 삭제</b>
<b>갈 색</b>	<b>부정확한 구속 조건</b>	<b>구속 조건의 삭제</b>



## IV. SKETCHER

### 4-1. Constraints Defined in Dialogue Box

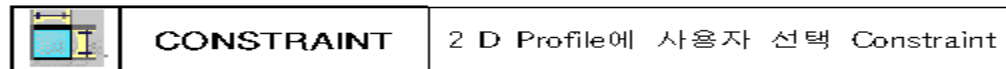
	<b>CONSTRAINTS VIA DIALOGUE BOX</b>	2D Profile에서 Constraint 값을 Dialogue Box에서 정의
--	---	--



- **Distance** : 두개의 Element 을 선택하여 거리 값을 부여
- **Length** : 하나의 Element 의 대한 길이 값을 부여
- **Angle** : 두개의 Element 을 선택하여 Angle 값을 부여
- **Radius / Diameter** : Circle / Arc 에 대한 값을 부여
- **Semimajor axis** : axis 을 정의
- **Semiminor axis** : axis 을 정의
- **Symmetry** : Symmetry 조건을 부여하는 기능
- **Midpoint** : Element 에 대한 Midpoint 을 부여
- **Equidistant point** : 등거리 점 조건을 부여하는 기능
- **Fix** : Element 에 대해 고정 시키는 조건을 부여
- **Coincidence** : 두개의 Element 을 일치시키는 조건 부여
- **Concentricity** : 두개의 Circle 을 중심을 맞추는 조건 부여
- **Tangency** : Arc/Circle 과 line 에 Tangency 조건 부여
- **Parallelism** : 두개의 Element 에 대한 평행조건을 부여
- **Perpendicularity** :
- **Horizontality** : 수평이라는 정의를 부여할 때 사용
- **Verticality** : 수직이라는 정의를 부여할 때 사용

# IV. SKETCHER

## 4-2. Constraints



■ **Constraint :**



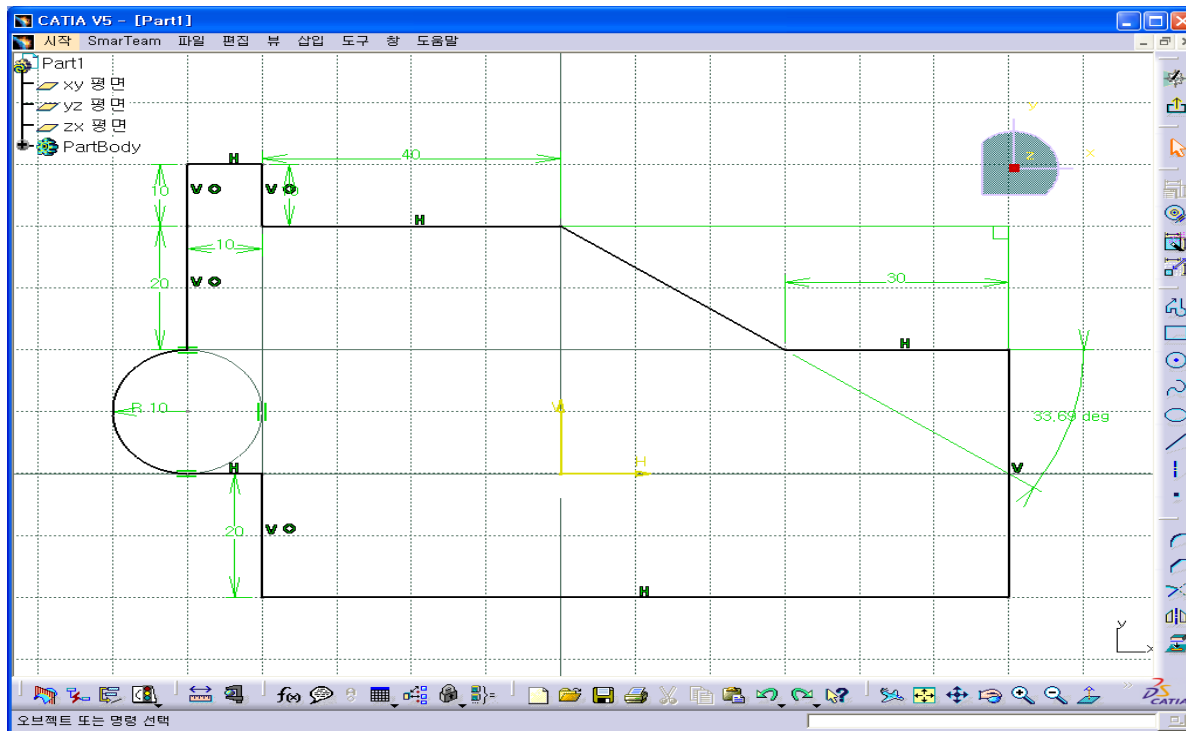
■ **Contact Constraint**



# IV. SKETCHER

## 4-3. Auto Constraint

	<b>AUTO CONSTRAINT</b>	자동으로 Constraint 부여
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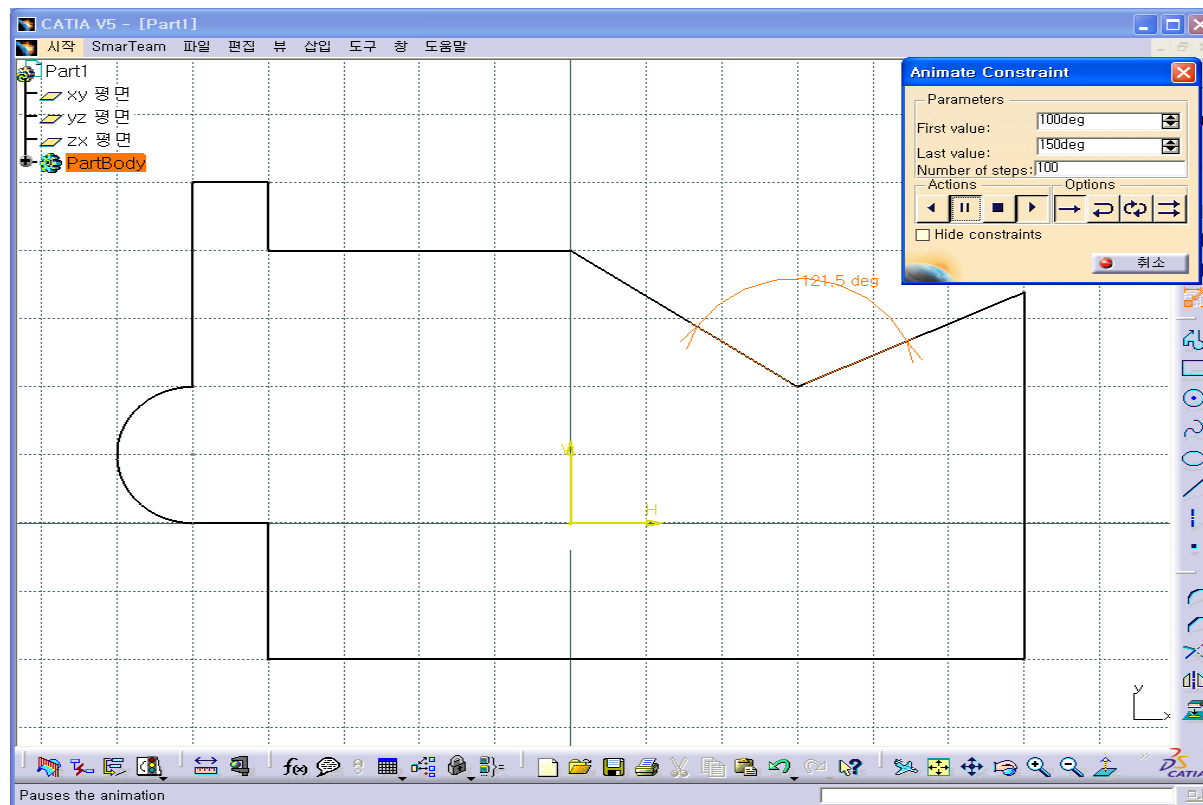
## IV. SKETCHER

### 4-4. Animate Constraint



**ANIMATE  
CONSTRAINT**

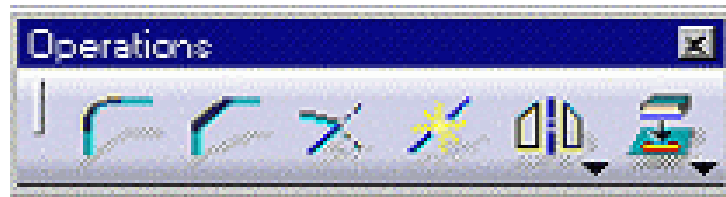
Constraint에 부여된 Value를 자동으로 조절하면서 최상의 Value를 생성



## IV. SKETCHER

### 5. Operations on Profiles

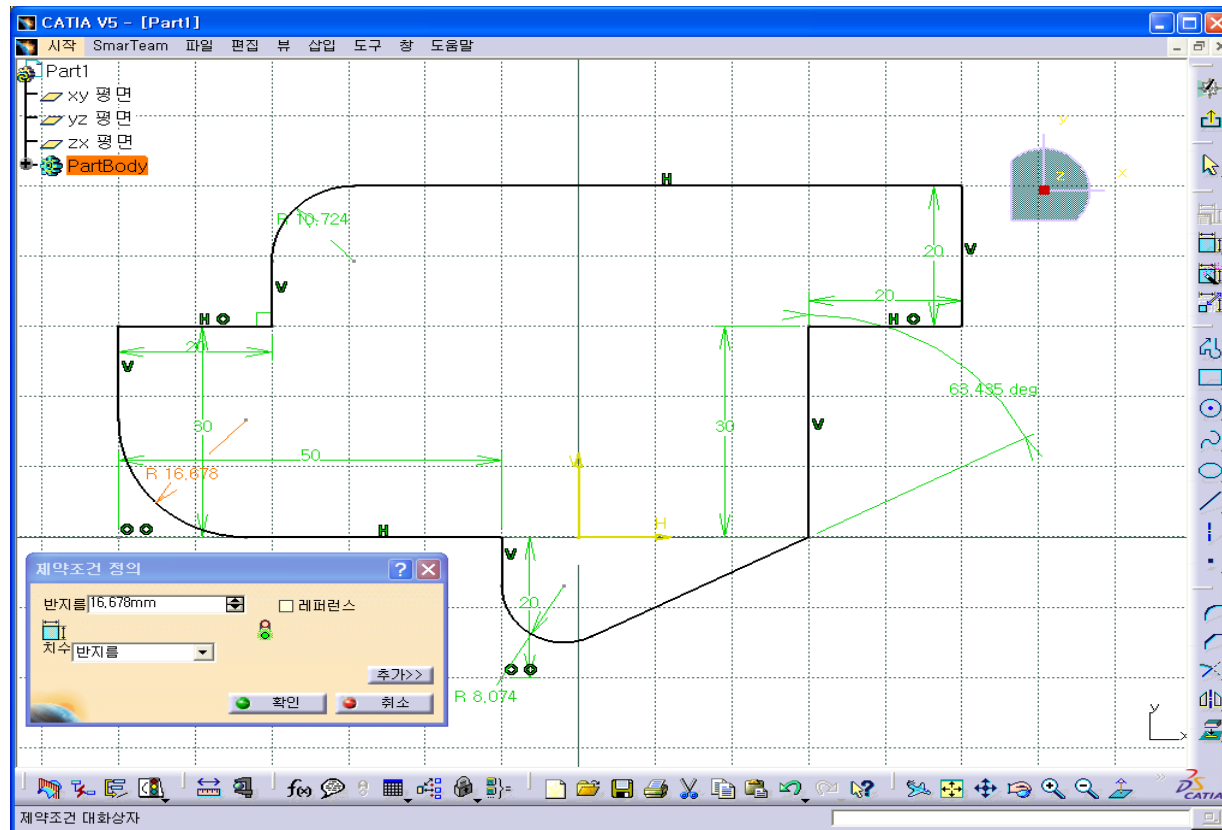
생성한 Profile 의 편집이나 이동, 복사 등의 기능을 모아  
놓은 툴 바이다.



# IV. SKETCHER

## 5-1. Corner

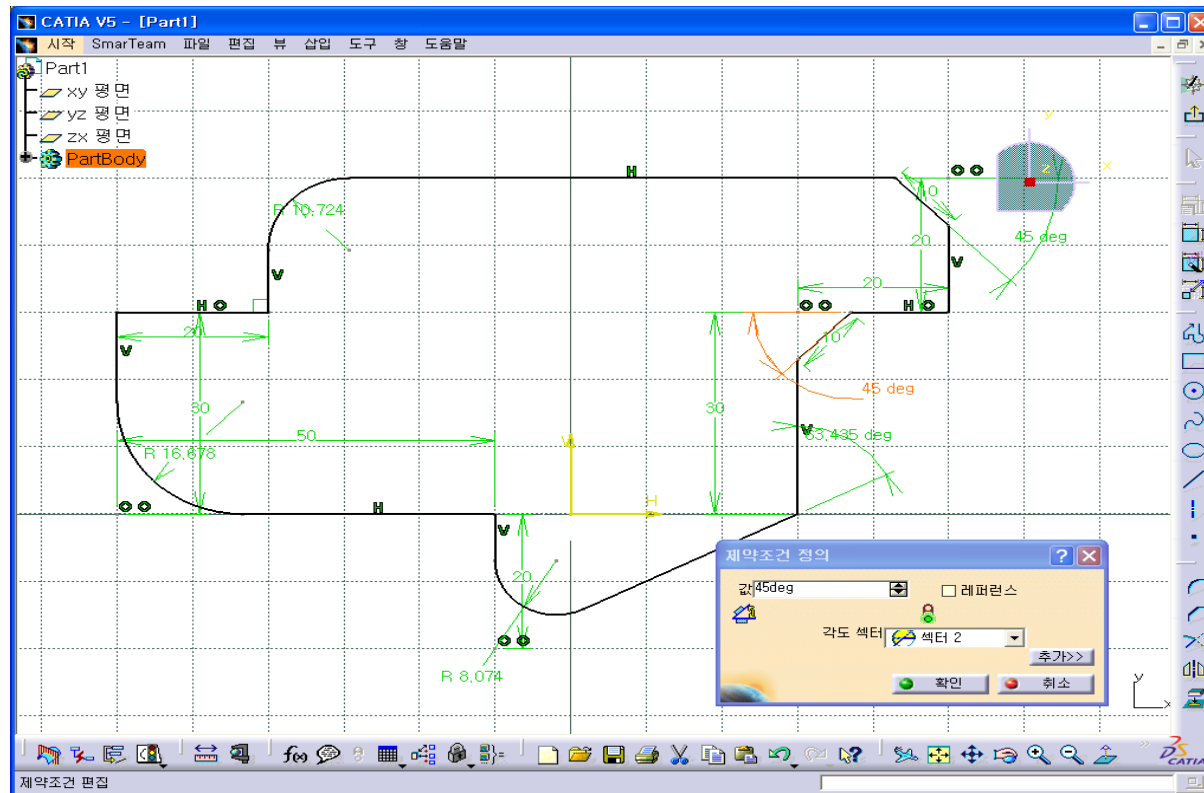
	<b>CORNER</b>	Corner 처리시 사용하는 기능
--	---------------	--------------------



# IV. SKETCHER

## 5-2. Chamber

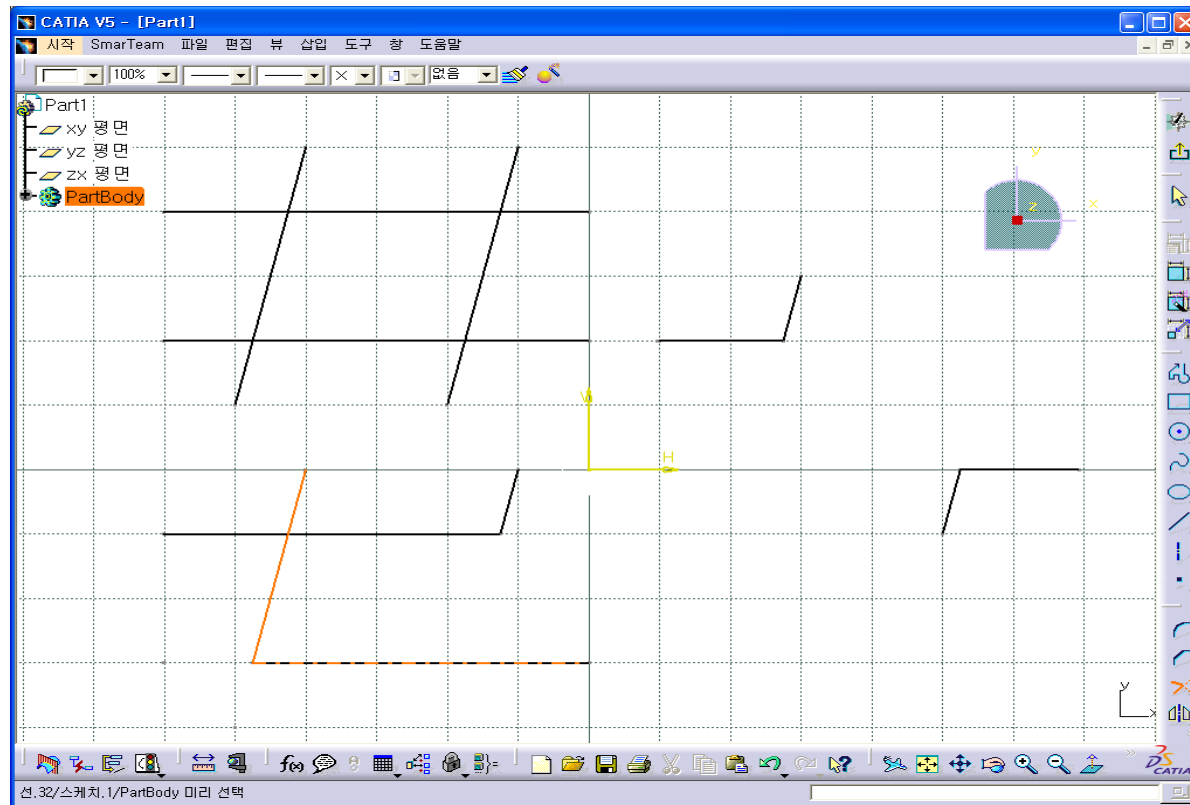
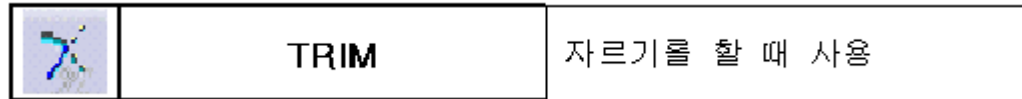
	<b>CHAMFER</b>	모따기 작업시 사용하는 기능
--	----------------	-----------------





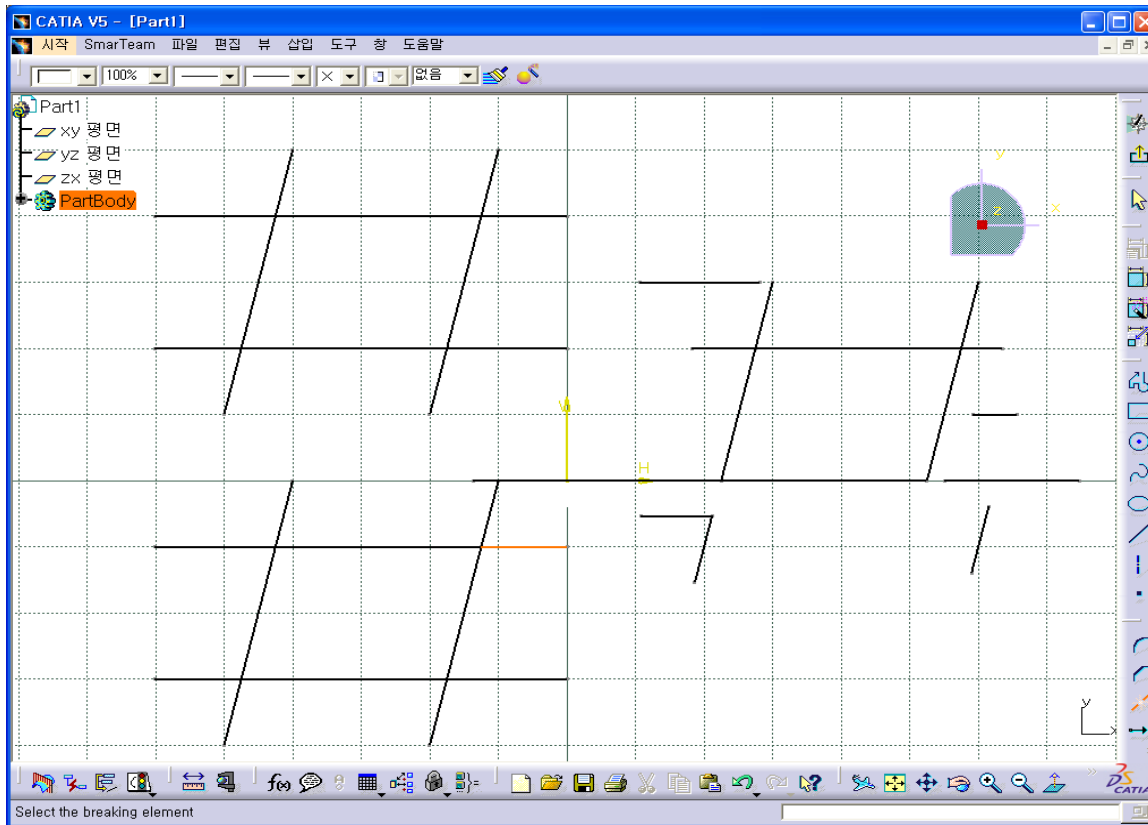
## IV. SKETCHER

### 5-3. Trim, Quick Trim



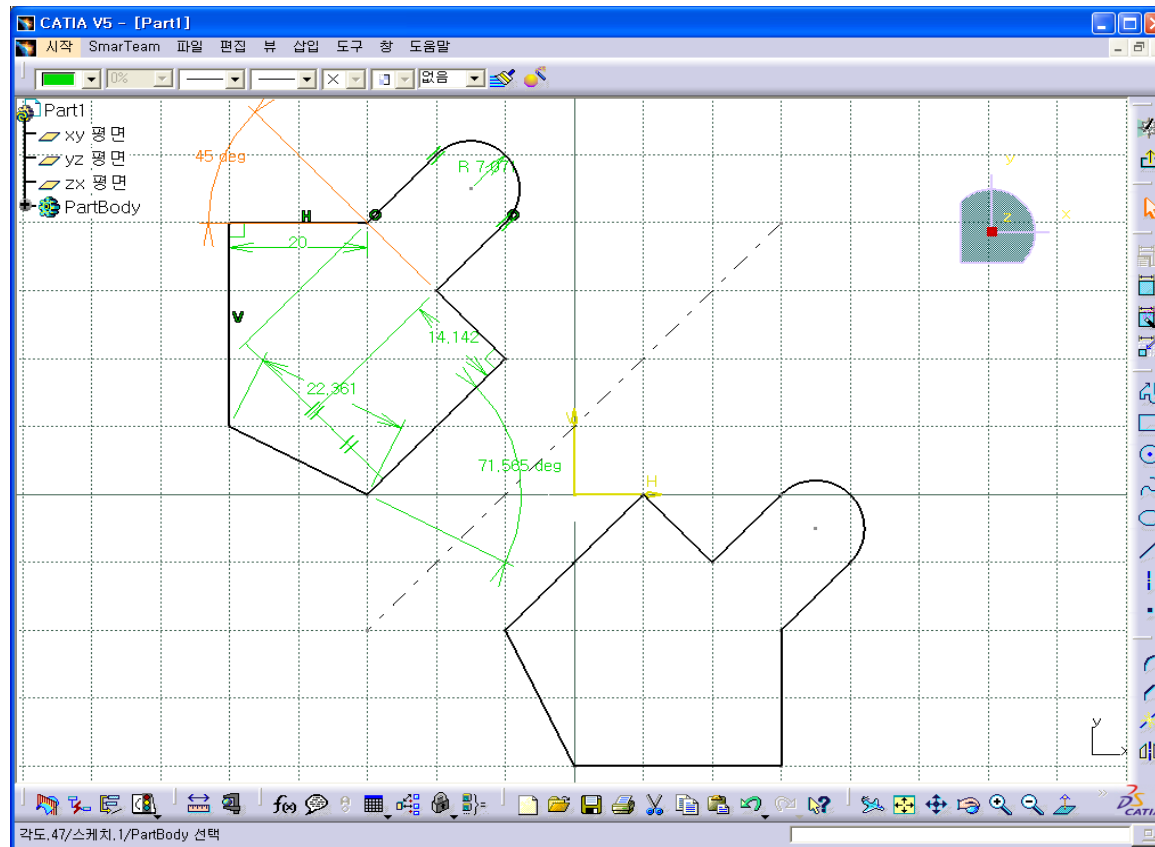
## IV. SKETCHER

### 5-4. Break, Close, Complement



## IV. SKETCHER

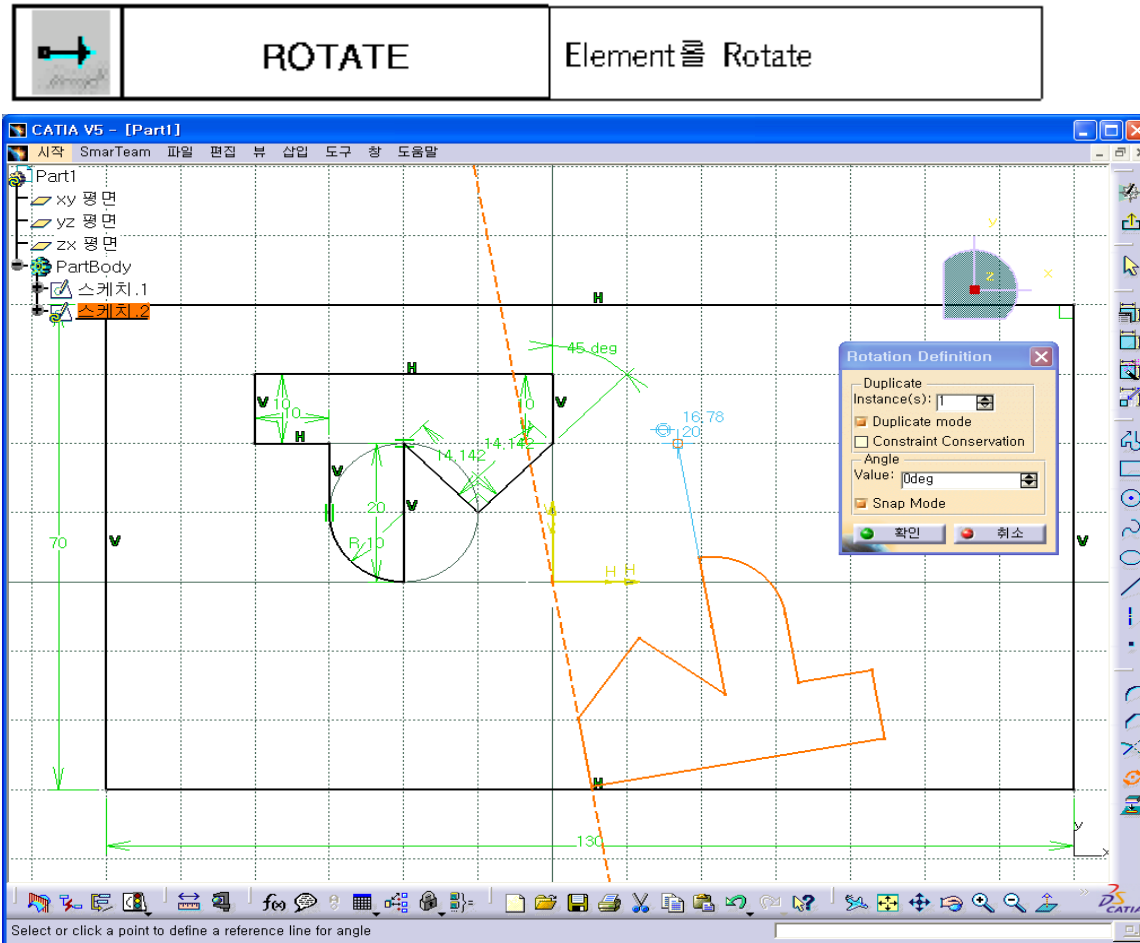
### 5-5. Mirror, Symmetry





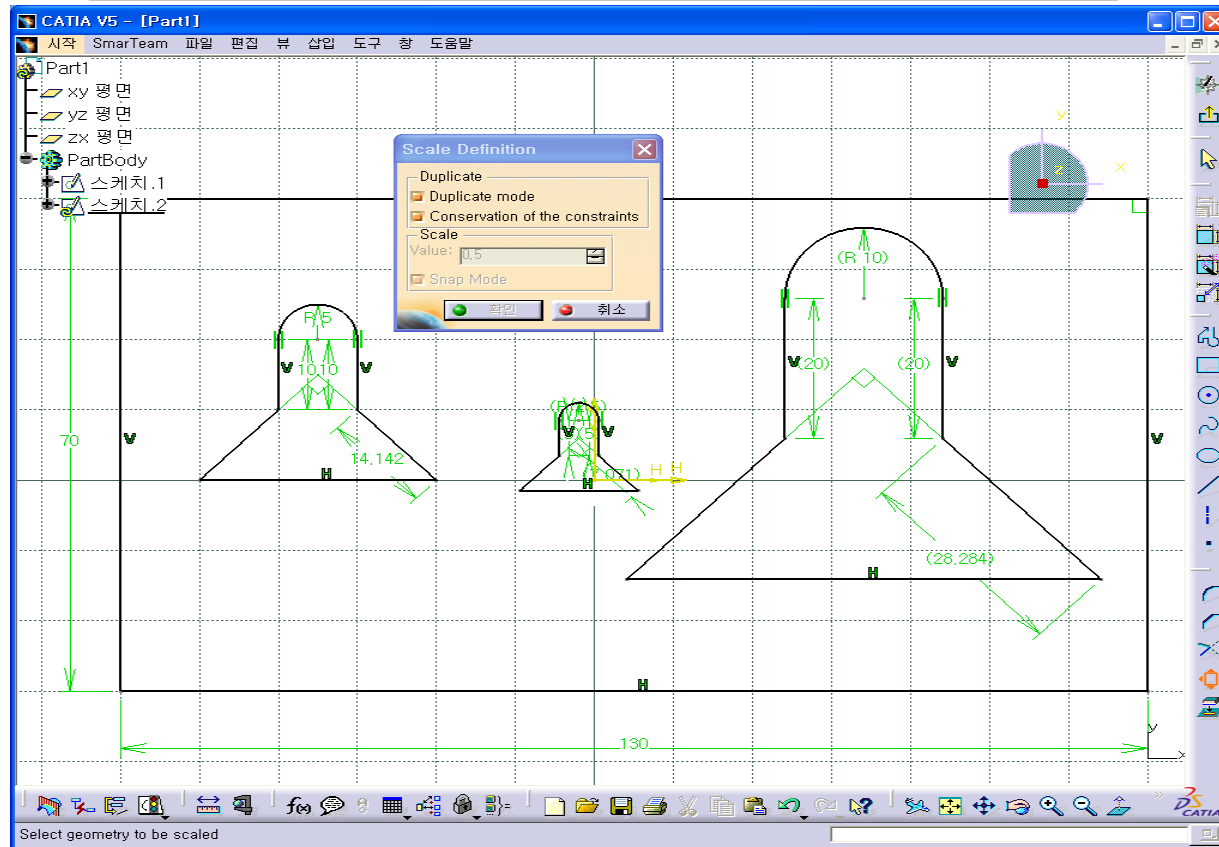
# IV. SKETCHER

## 5-7. Rotate



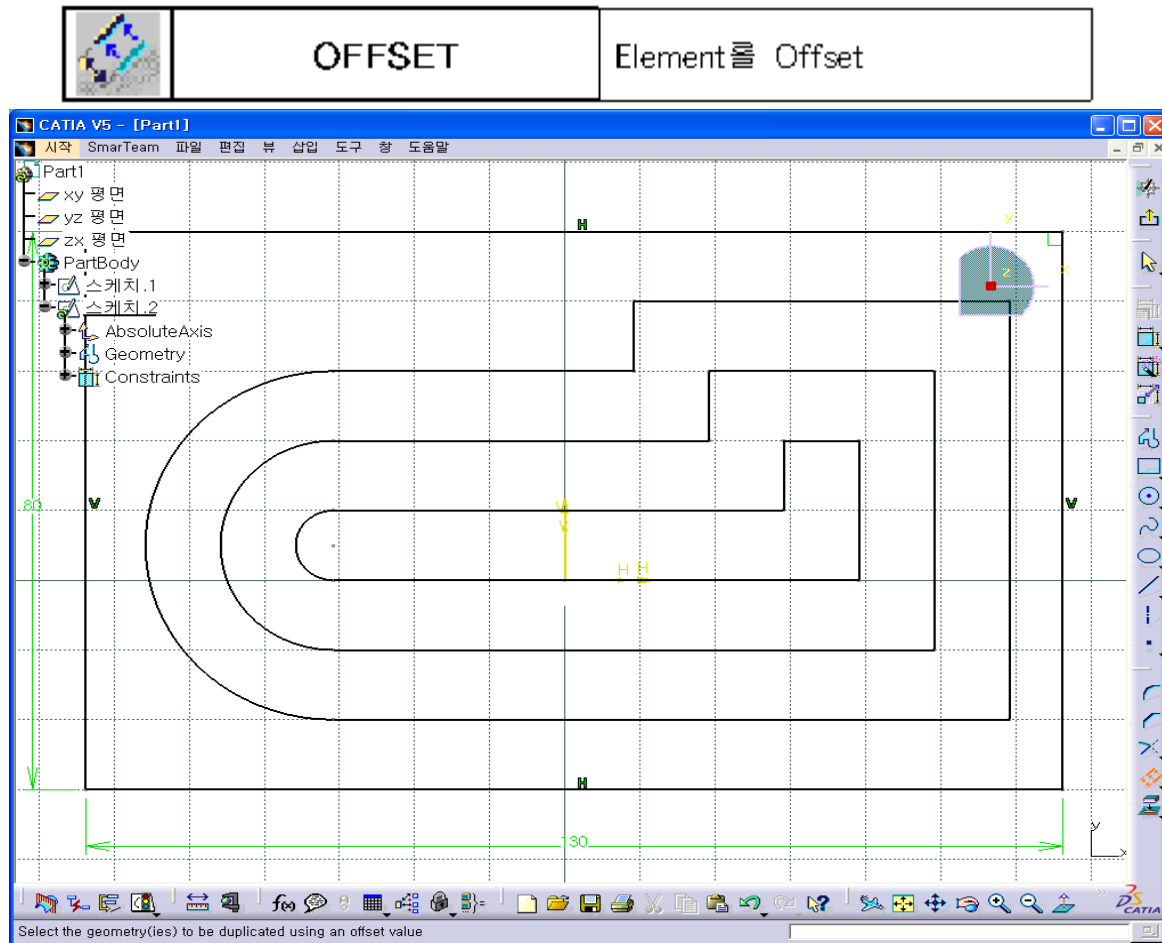
# IV. SKETCHER

## 5-8. Scale



## IV. SKETCHER

### 5-9. Offset






## IV. SKETCHER


### 5-10. Project 3D Elements

	PROJECT 3D ELEMENT	Plane 상에 3D Element를 투영
---	--------------------	-------------------------

### 5-11. Intersect 3D Elements

	INTERSECT 3D ELEMENT	Plane 상에서 3D Element들의 Intersect element <u>를</u> 생성
---	-------------------------	---

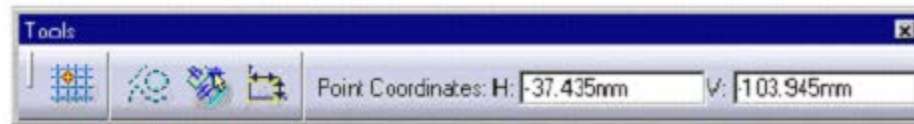
### 5-12. Isolate





	ISOLATE	isolating
---	---------	-----------

## IV. SKETCHER

### 6. Tools

Tools 가 가지는 기본 기능 외에 다른 아이콘의 보조적인 기능을 생성하기도 하고 Profile 의 좌표값 이나 크기 등을 나타내거나 조정 할 수 있는 툴 바이다.



-  Snap to Point
-  Construction/Standard Element
-  Geometrical Constraints
-  Dimensional Constraints

## IV. SKETCHER

### 6-1. Snap to point

Sketcher 의 바탕화면에 나타나는 격자모양의 교점사이로 마우스 Pointer를 움직이게 해준다

### 6-2. Construction/Standard Element

일반적인 Profile을 생성할 때 생성되는 Line 이 Standard Element 이고 또 하나는 파선으로 생성되는 Construction Element 이다.  
Construction Element 는 Standard Element Profile을 생성할 수 있도록 도와주는 보조 Profile 이다.

### 6-3. Geometrical Constraints

치수와 관계된 구속조건들을 생성 시켜준다.

### 6-4. Dimensional Constraints

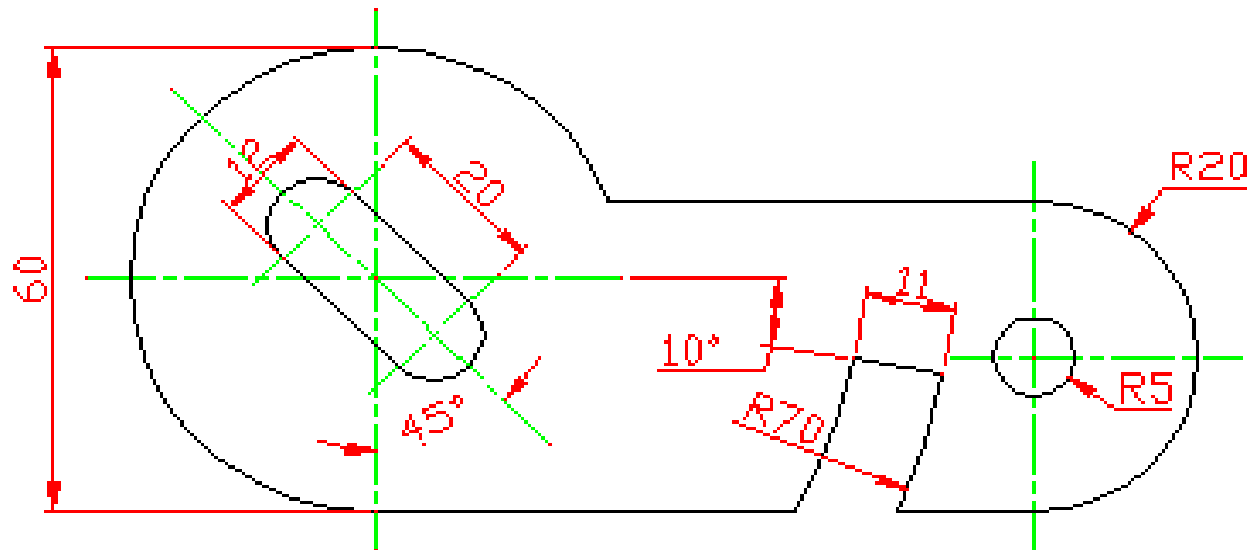
치수와 관계된 구속조건들을 생성 시켜준다.



## IV. SKETCHER

## 7. 예 제도면 (2)

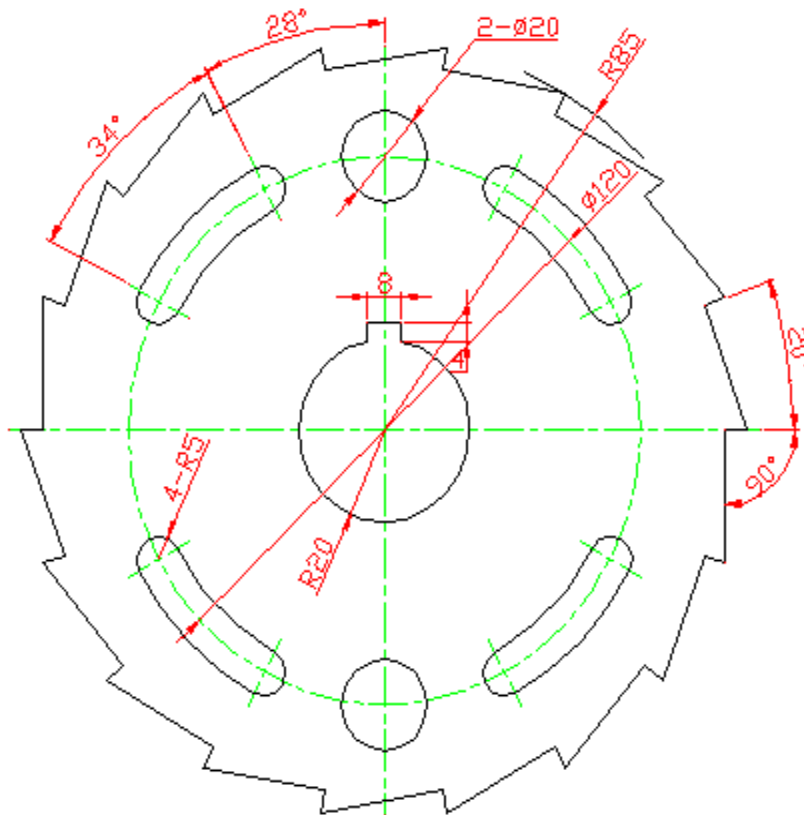
## ARC, 상대극 좌표



## IV. SKETCHER

### 7. 예 제도면 (3)

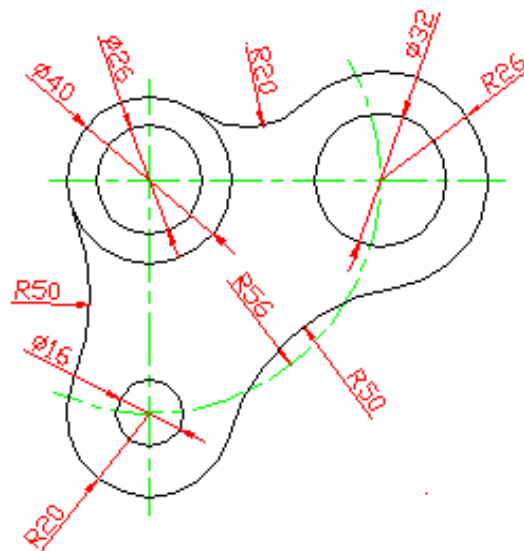
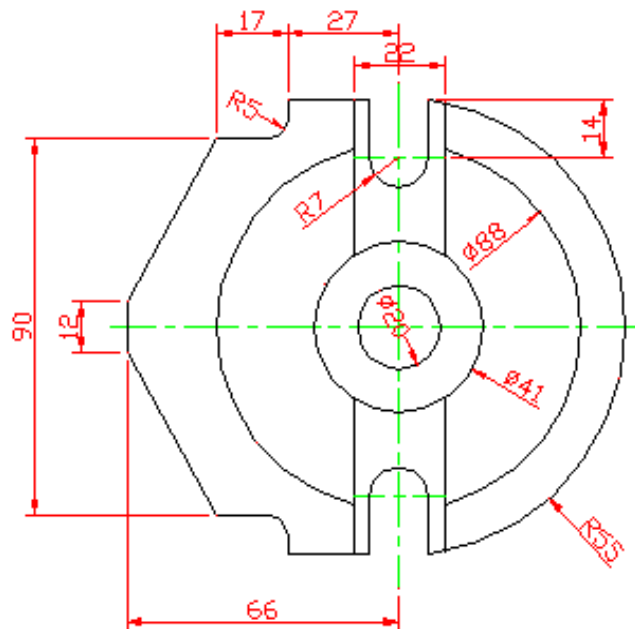
Array, 상대극좌표, Mirror



## IV. SKETCHER

### 7. 예 제도면 (4)

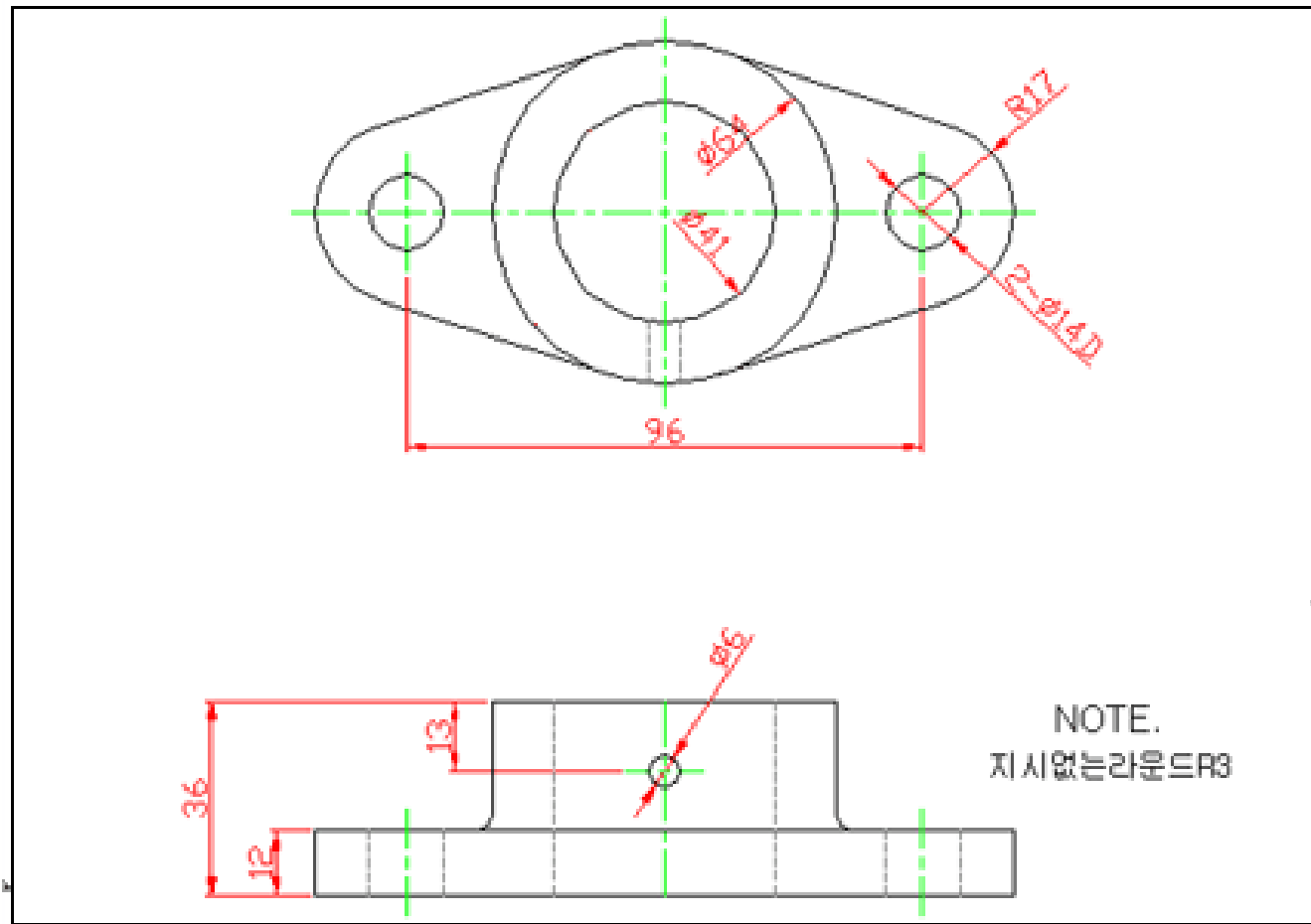
MIRROR 및 CIRCLE, FILLET 응용





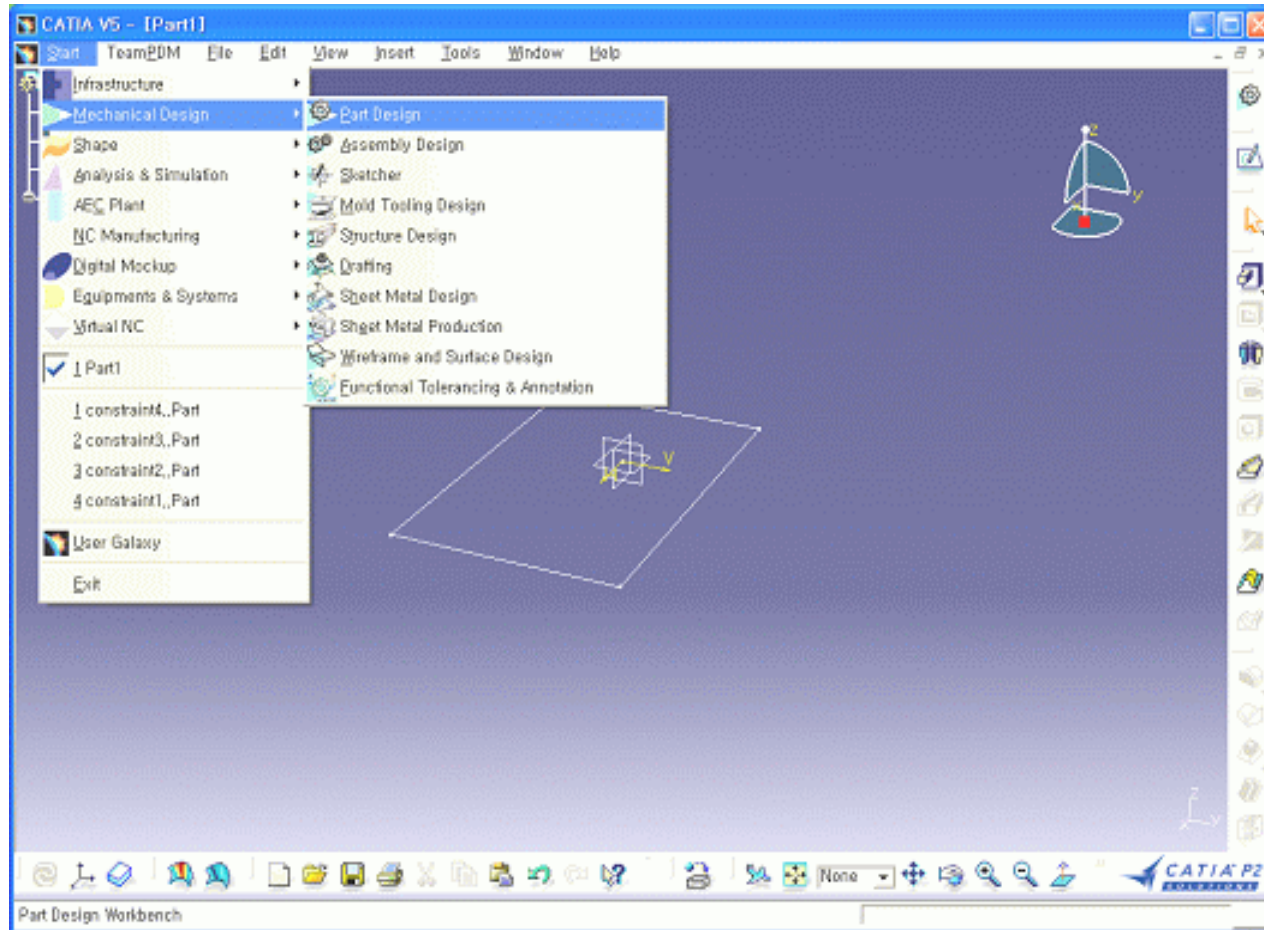
## IV. SKETCHER

### 7. 예 제도면 (5)



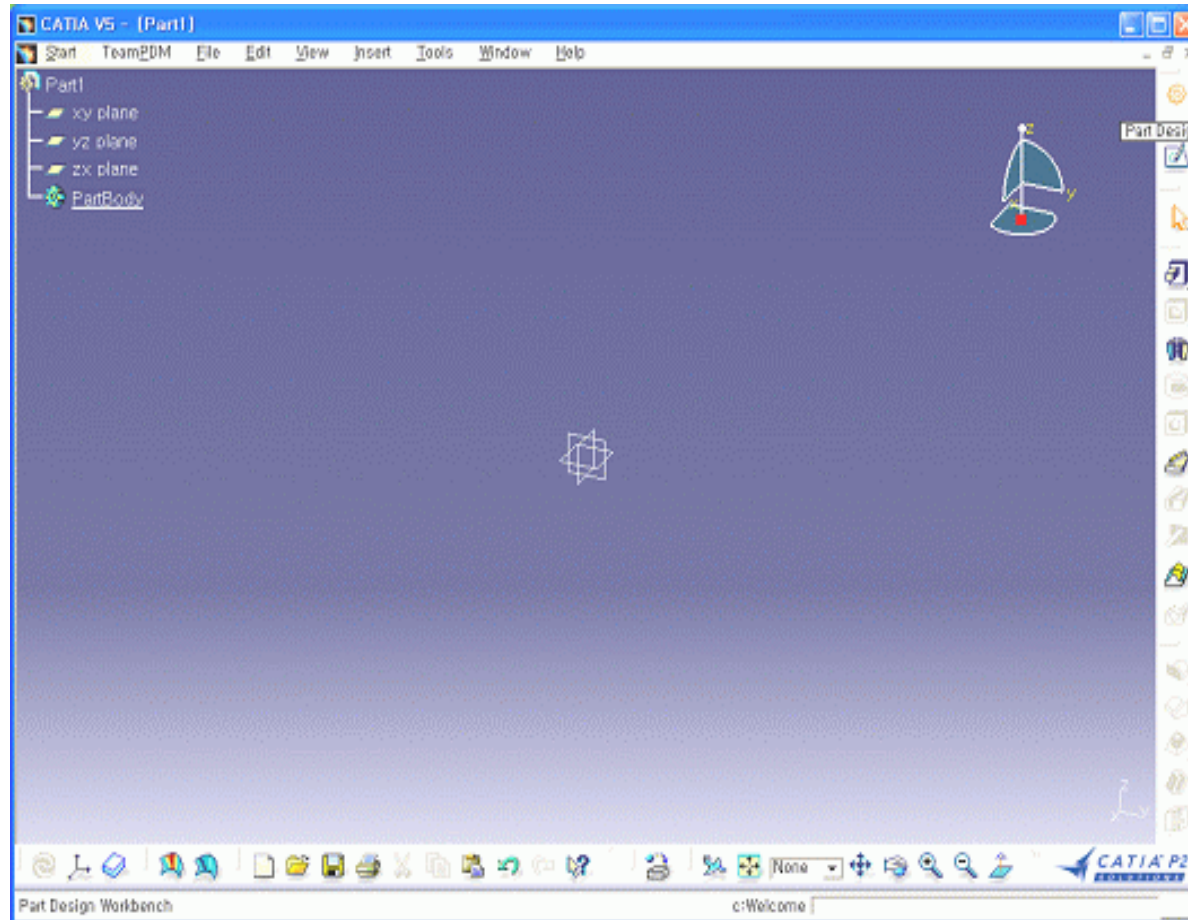
# V. PART DESIGN

## 1. Introduction (1)



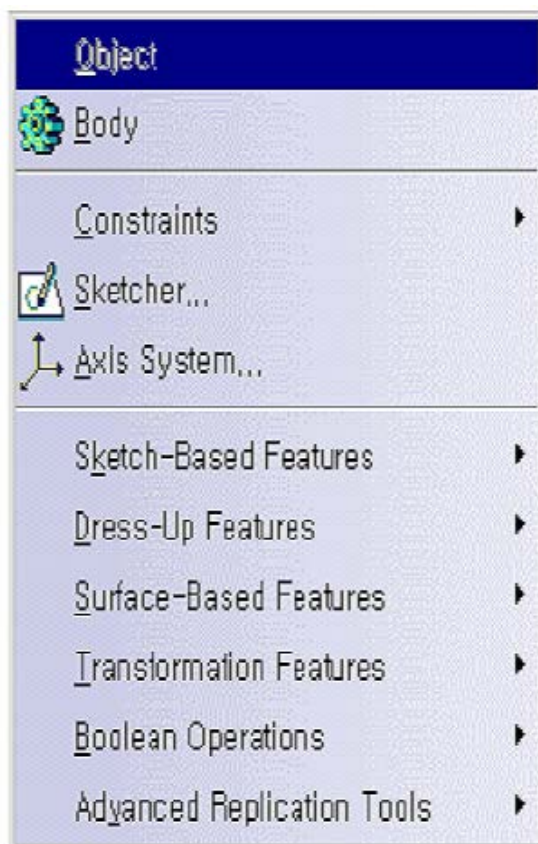
# V. PART DESIGN

## 1. Introduction (2)



# V. PART DESIGN

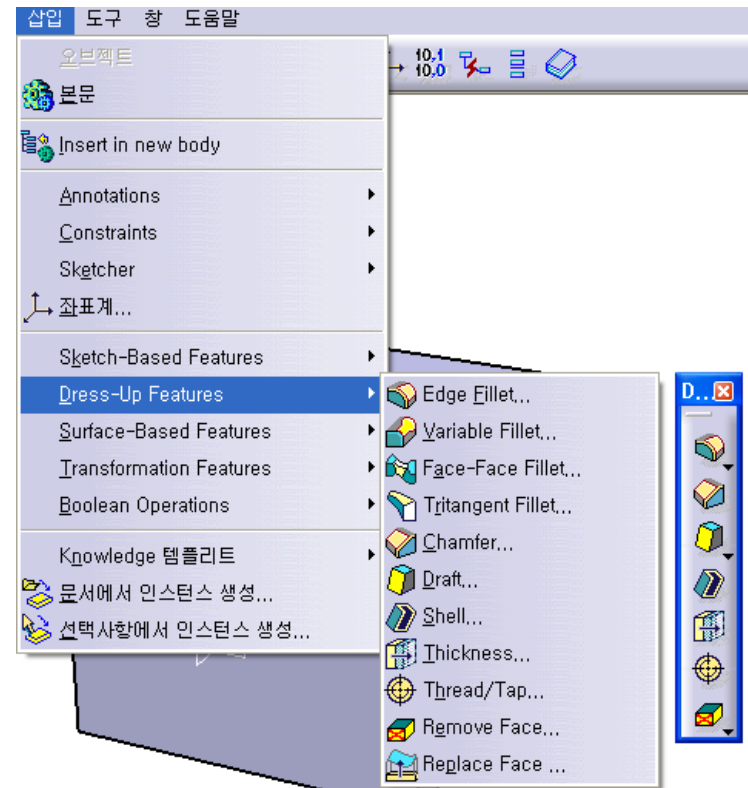
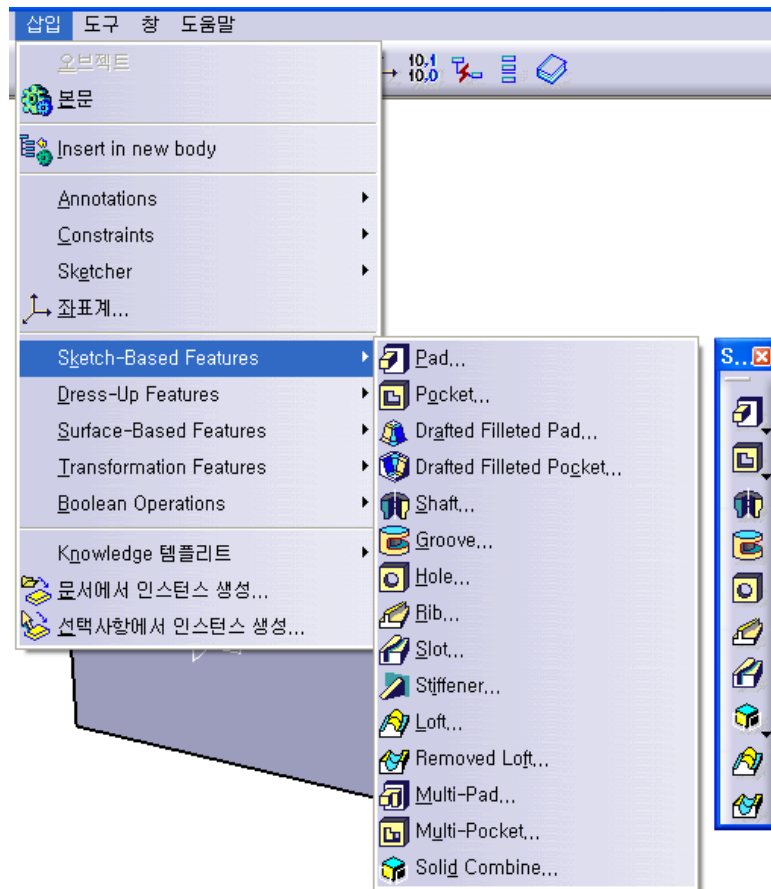
## 2. Menu bar 와 Icon (1)



- **Body** : 새로운 Part Body 를 생성할 때 사용하는데 하나의 CATPart 파일에는 여러 개의 Body 가 존재할 수 있다.
- **Constraint** : 2D Profile 에 Parameter 를 부여할 때 사용한다.
- **Reference Element** : Plane(Offset, Plane With Angle), Line, Point 등을 생성할 때 사용한다.
- **Sketcher...** : 2D Profile 작업을 위한 Mode 로 전환할 때 사용한다.
- **Axis** : 상대 축을 생성하는 기능 (V4 Axis 와 동일)
- **Sketch-based Feature** : Pad, Pocket, Hole, Shaft, Groove, Stiffener, Rib, Slot 등을 생성할 때 사용한다.
- **Dress-up Feature** : Fillet, Draft, Shell, Thickness, Chamfer 등을 생성할 때 사용한다.
- **Surface Based Feature** : Split, Thick Surface, Close Surface, Sew Surface 등으로 구성되어 있다.(V4 의 Complex Solid)
- **Transformation Feature** : Pattern, Mirror, Split, Scaling 등을 생성할 때 사용한다.
- **Boolean Operations** : Assembly, Add, Remove, Intersect, Union trim, Remove Lump 로 구성
- **Advanced replication Tools** :

# V. PART DESIGN

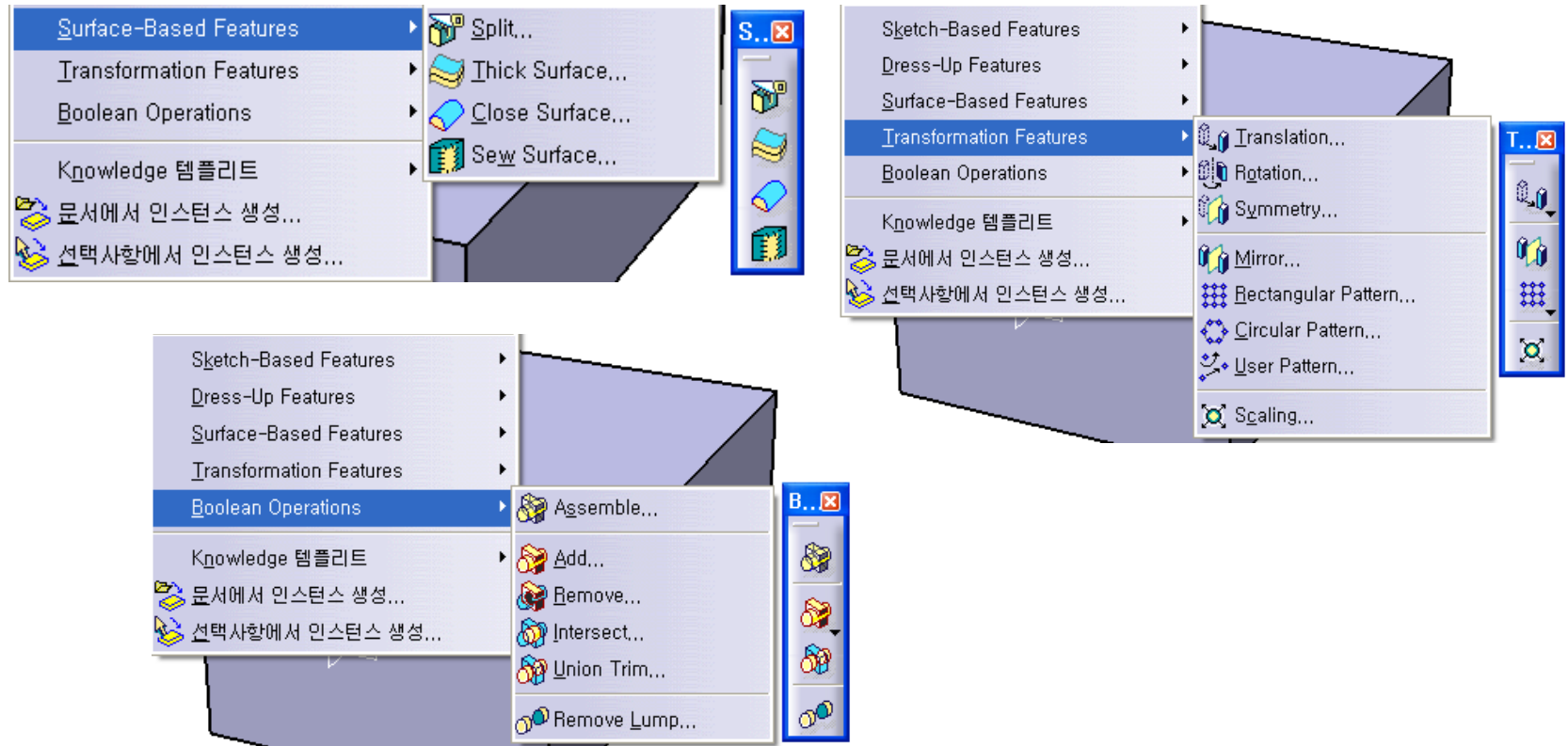
## 2. Menu bar 와 Icon (2)





# V. PART DESIGN

## 2. Menu bar 와 Icon (3)



## V. PART DESIGN

### 3. Sketch Based Features

Sketcher에서 만든 Profile 을 솔리드로 생성하기 위한 가장 기본적인 작업 툴 바



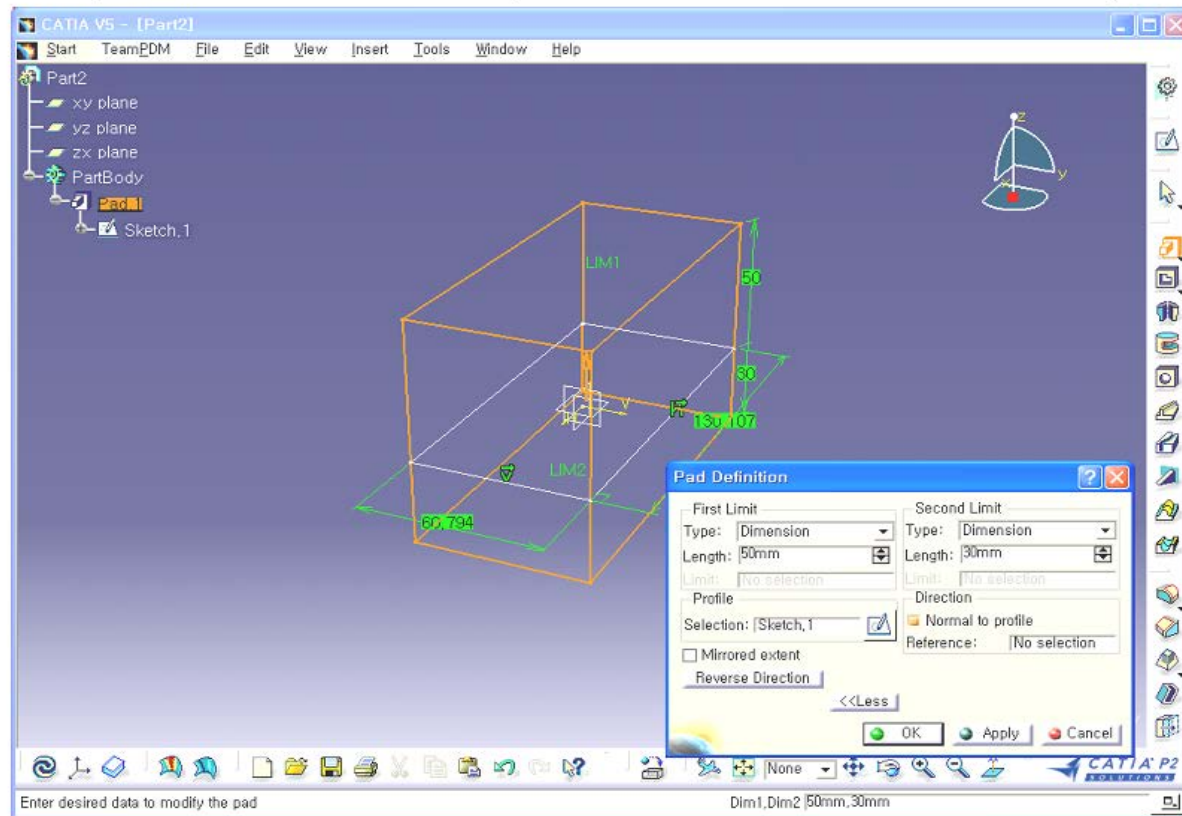
# V. PART DESIGN

## 3-1. Pad



PAD

2 D Profile을 솔리드로 작성





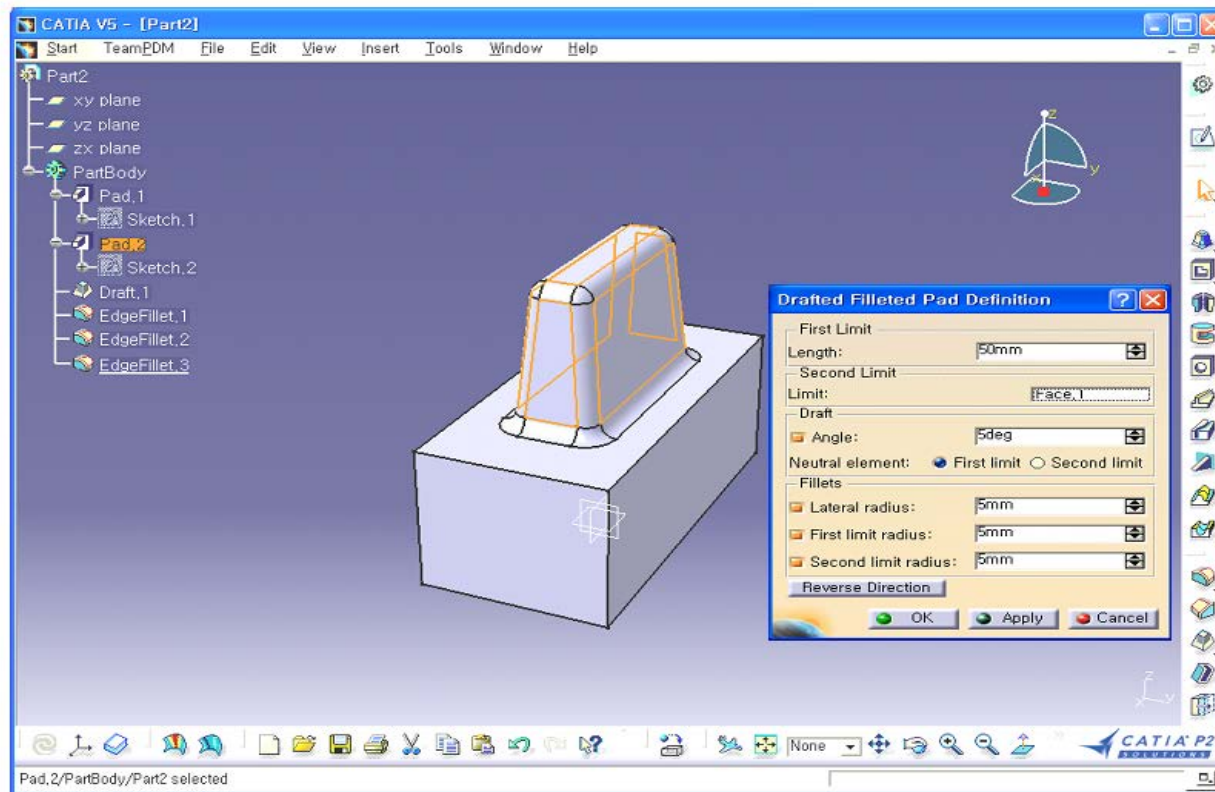
# V. PART DESIGN

## 3-2. Drafted Filleted Pad



**DRAFTED  
FILLETED PAD**

Solid에 Drafting 과 Filleting 작업을  
동시에 수행 Pad 형 Feature 생성



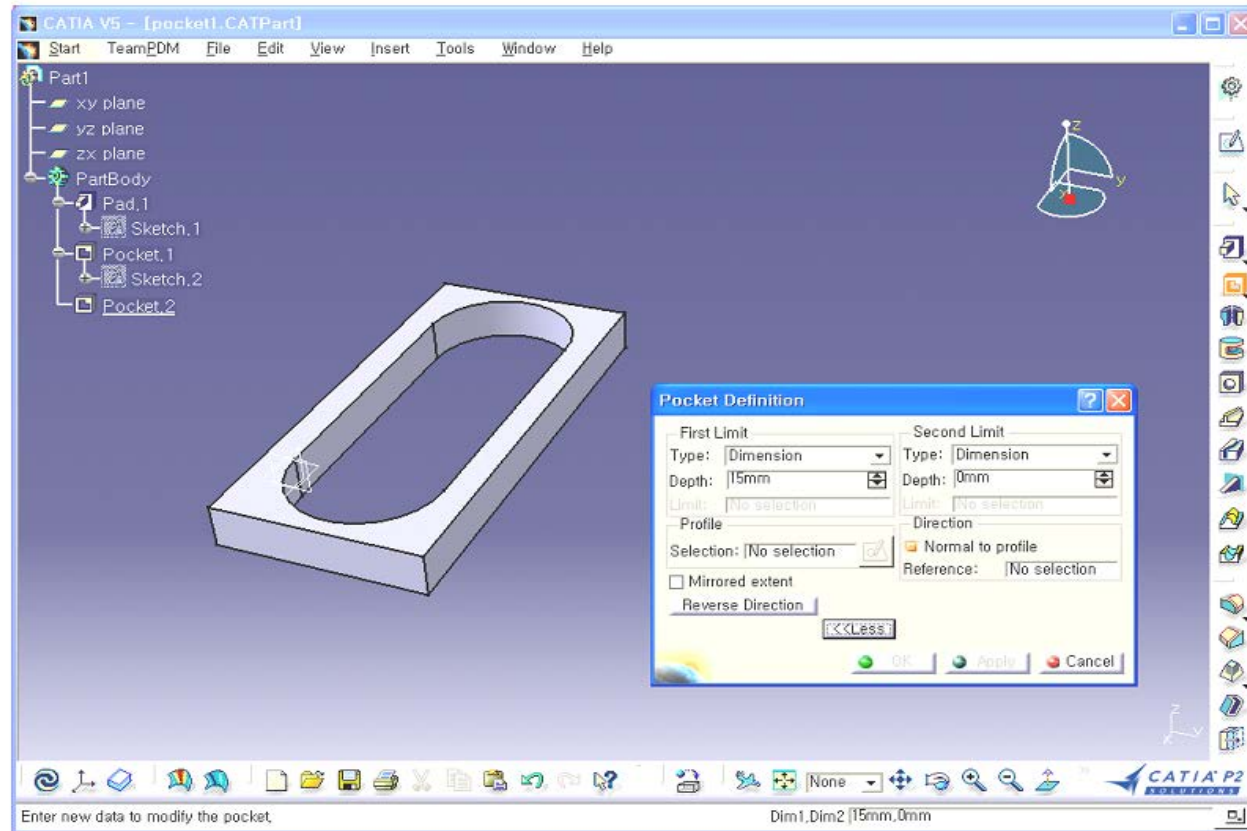
# V. PART DESIGN

## 3-3. Pocket



POCKET

솔리드에 Pocket 생성



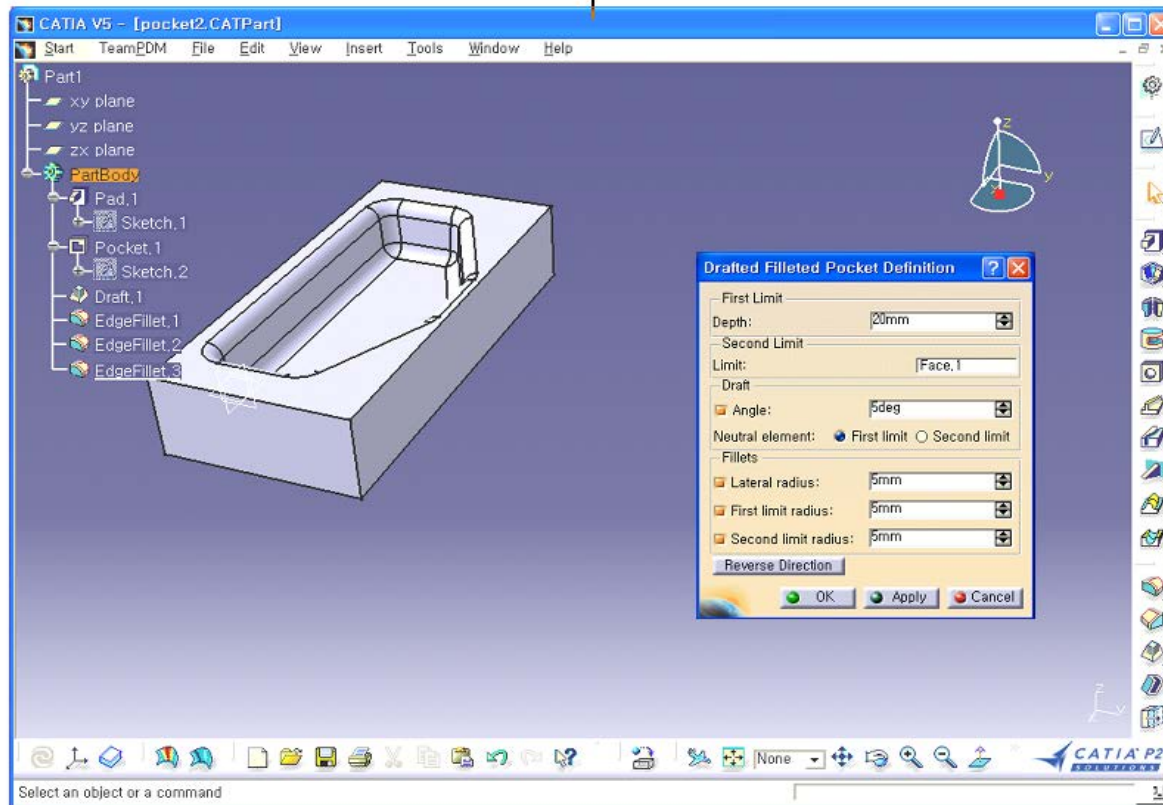
# V. PART DESIGN

## 3-4. Drafted Filleted Pocket




**DRAFTED FIL-  
LETED POCKET**

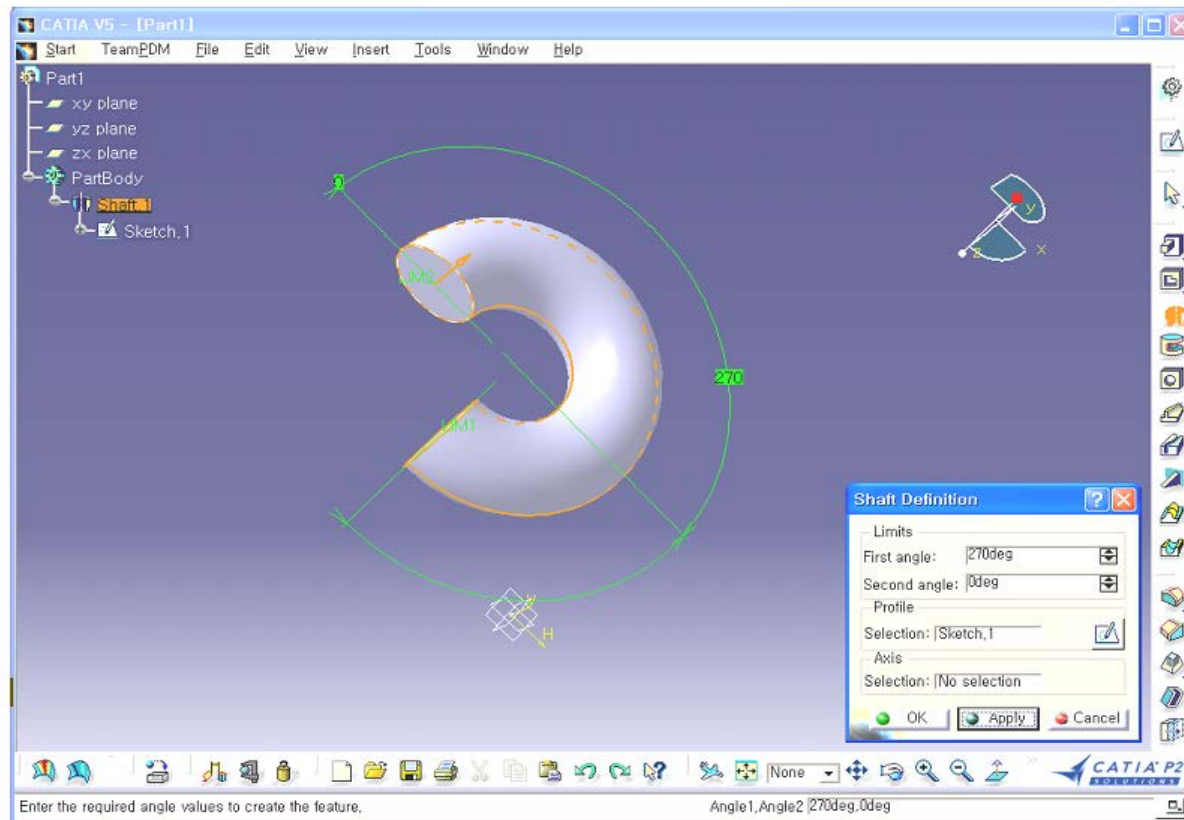
솔리드에 Drafting 과 Filleting 작업을 동  
시에 수행하면서 Pocket 형 Feature 생성



# V. PART DESIGN

## 3-5. Shaft

	SHAFT	회전형 Solid 형상을 생성하는 기능
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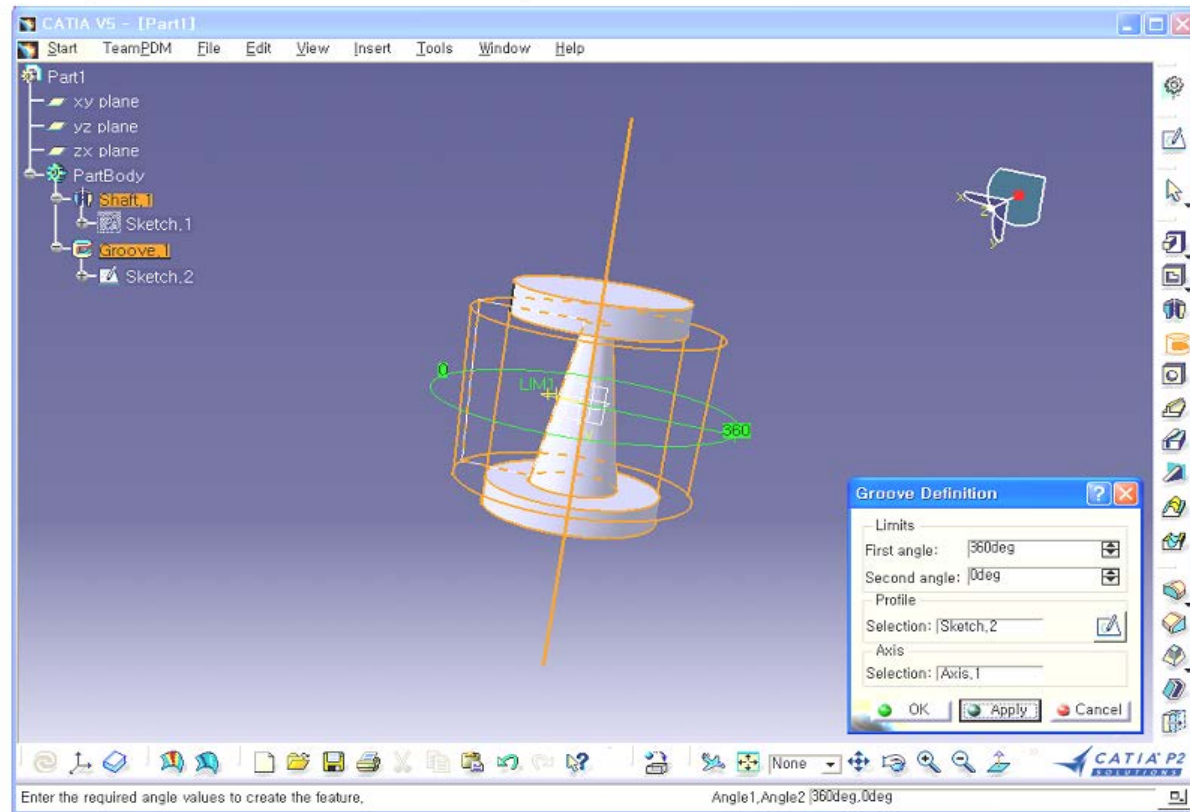
## V. PART DESIGN

### 3-6. Groove



GROOVE


회전형 Solid 형상을 생성하는 기능

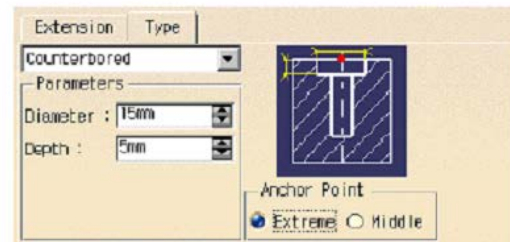
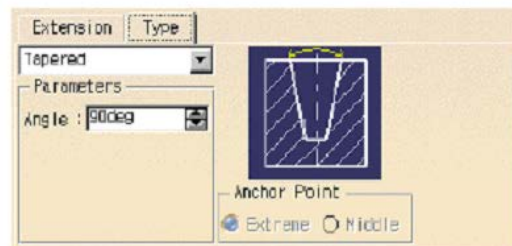
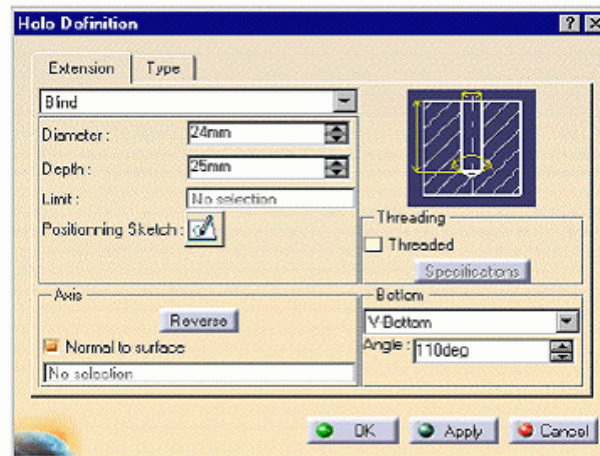




# V. PART DESIGN

## 3-7. Hole (1)

	<b>HOLE</b>	정의되어 있는 형상을 이용하여 구멍 뚫는 작업을 수행
---	-------------	-------------------------------



### ➤ Tapered Type

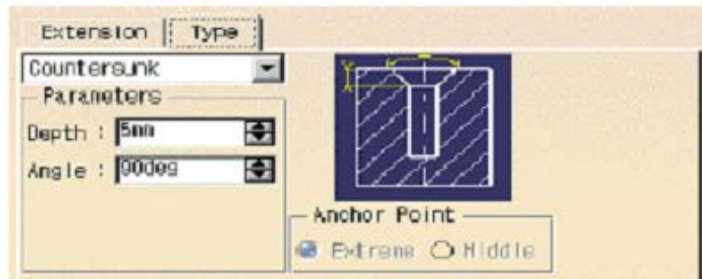
- Parameter / Anchor Point

### ➤ Counterbored Type

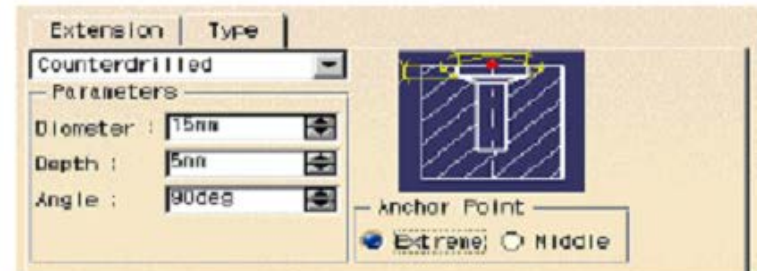
- Parameter / Anchor Point

# V. PART DESIGN

## 3-7. Hole (2)



- **Countersunk Type**  
- Parameter / Anchor Point



- **Counterdrilled Type**  
- Parameter / Anchor Point



Blind



Up to Next



Up to Last



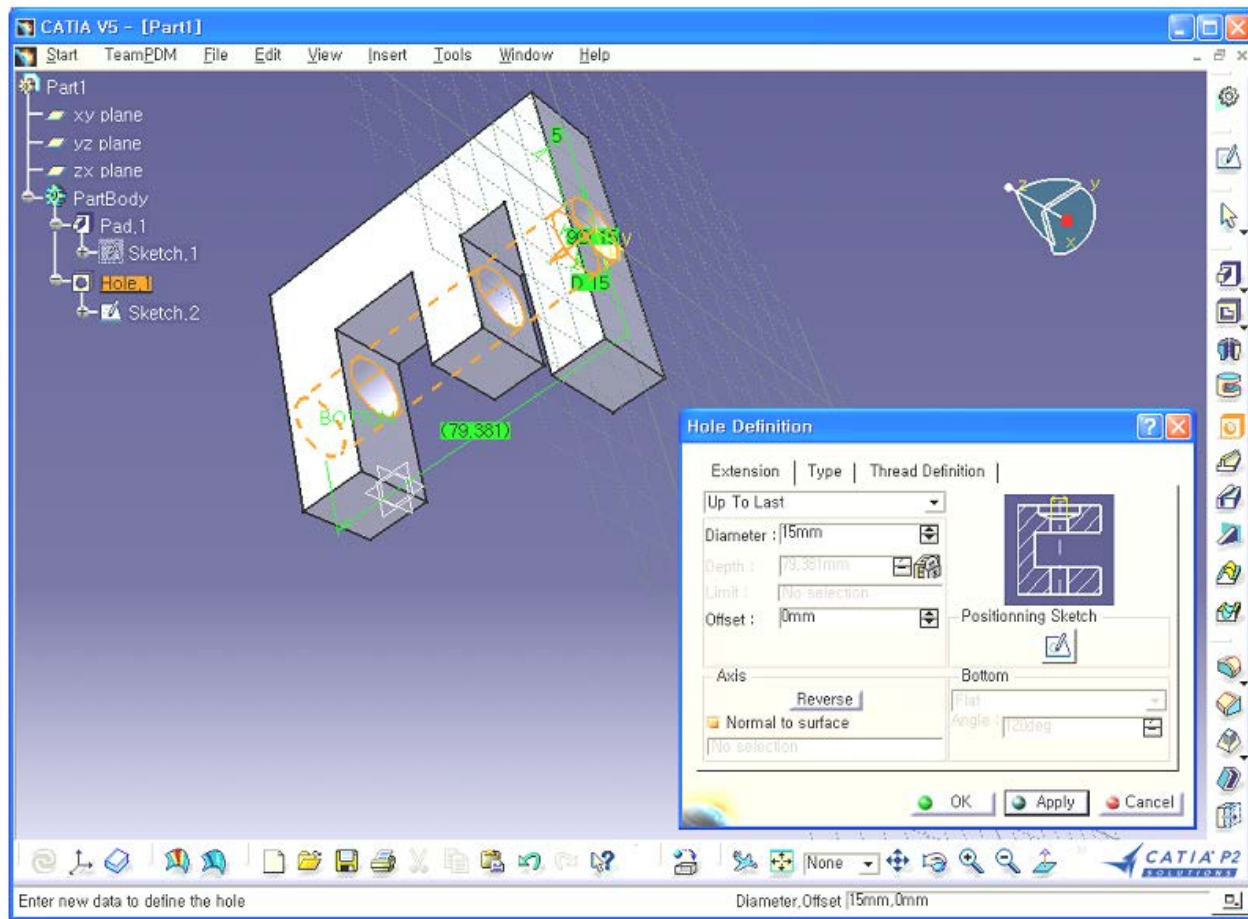
Up to Plane



Up to Surface

# V. PART DESIGN

## 3-7. Hole (3)





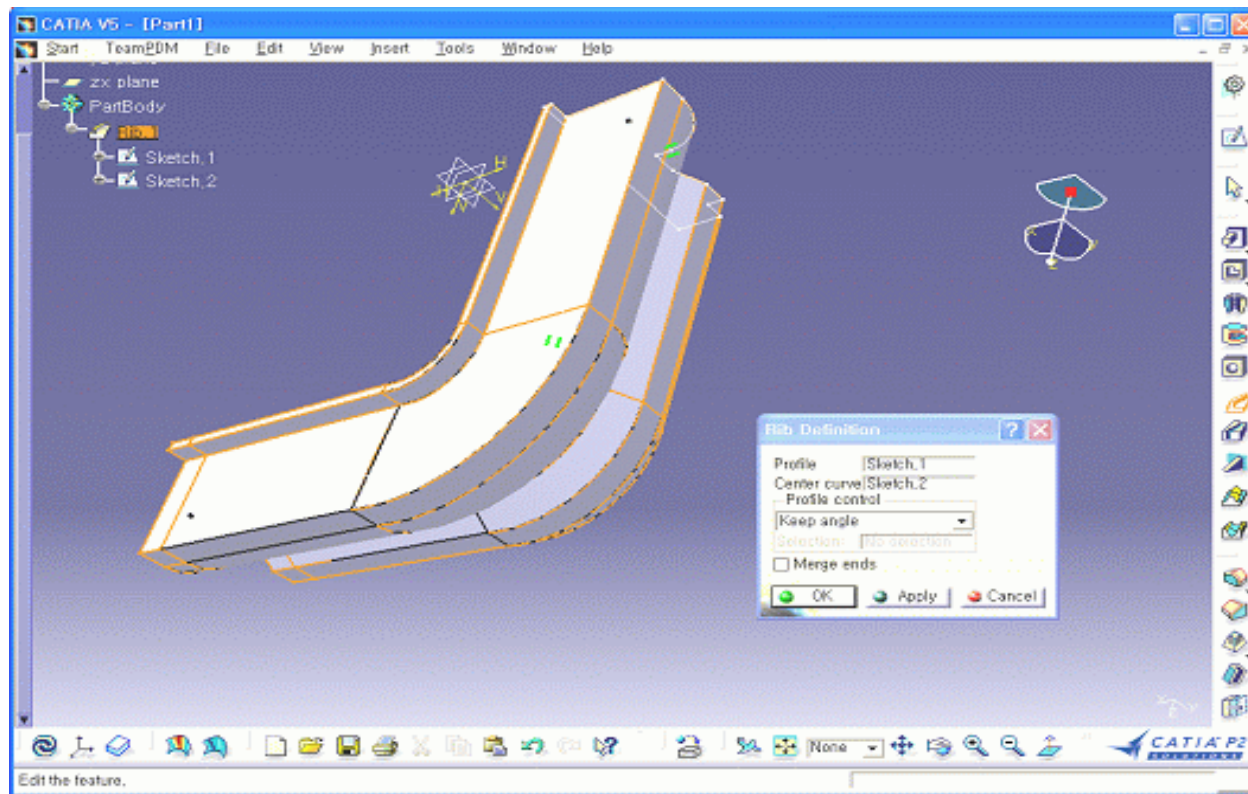
# V. PART DESIGN

## 3-8. Rib



RIB

Profile을 center curve를 따라 가는  
solid 형상을 생성하는 기능



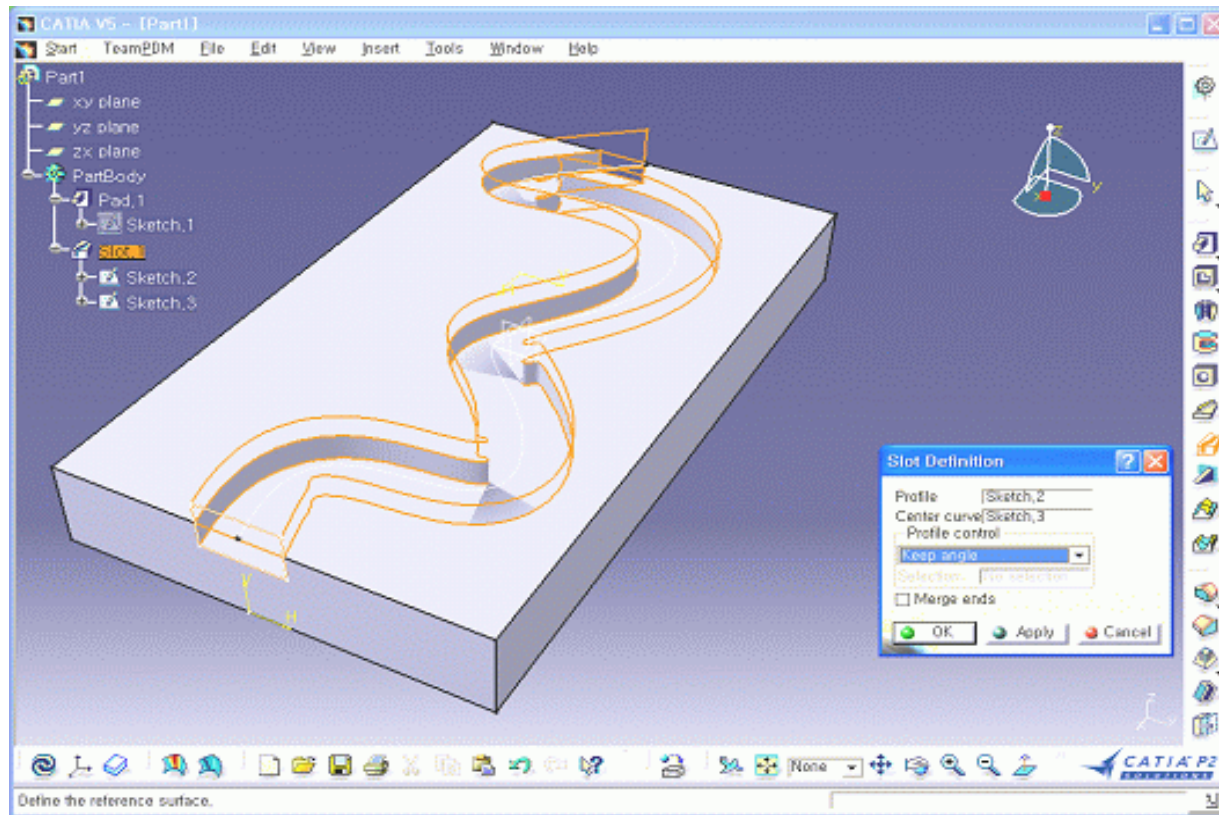
# V. PART DESIGN

## 3-9. Slot



SLOT

Profile을 center curve를 따라 가는  
solid 형상을 Pocket하는 기능



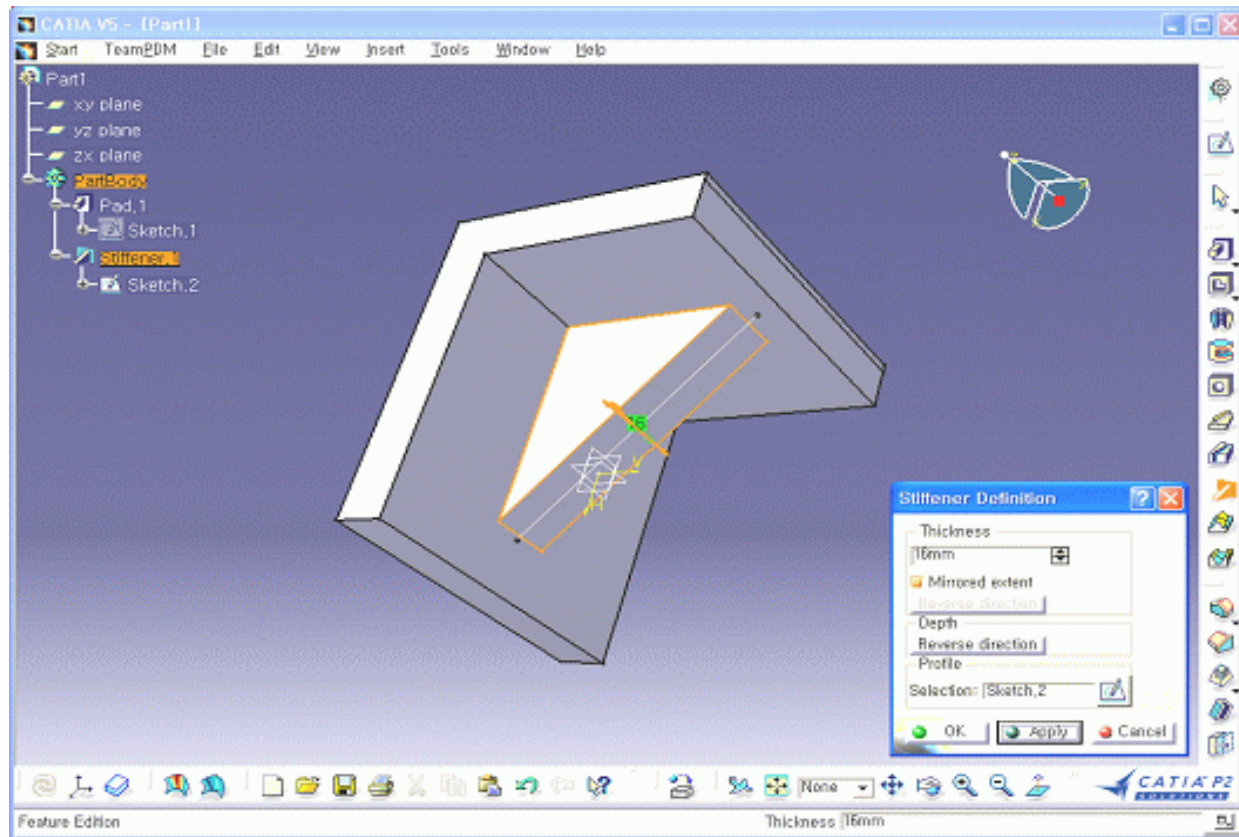
## V. PART DESIGN

### 3-10. Stiffener




STIFFENER

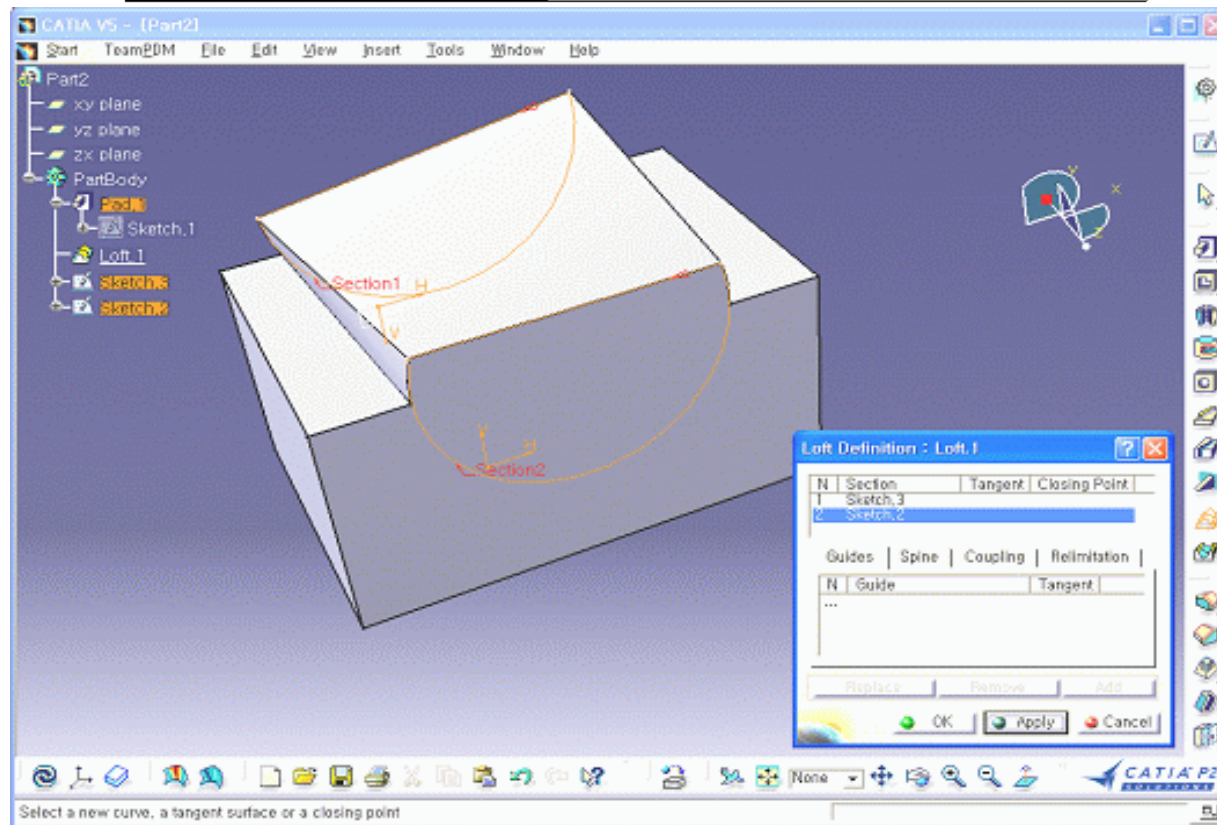
제품의 형상에 보강재를 붙일때



# V. PART DESIGN

## 3-11. Loft

	LOFT	Loft 형 Solid를 생성 (양의 곡성)
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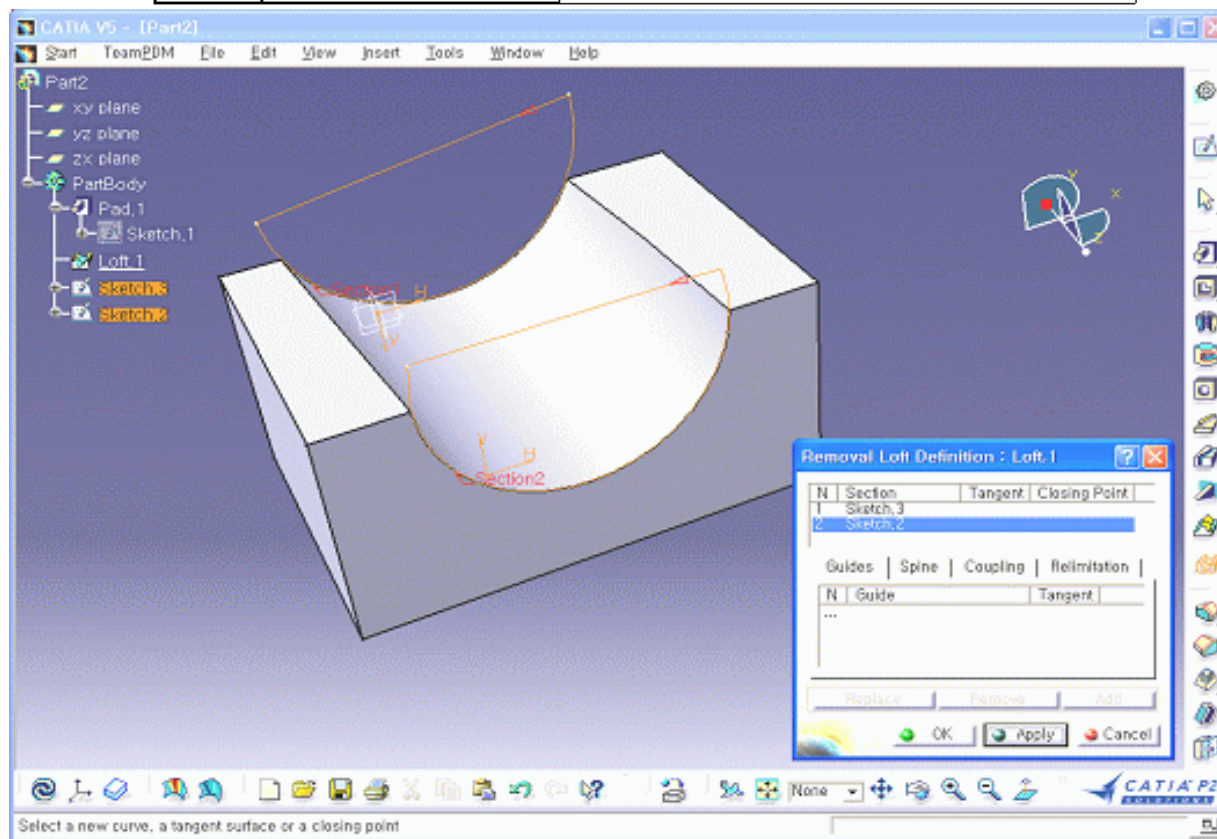
# V. PART DESIGN

## 3-12. Removed Loft



LOFT

Loft 형 Solid를 생성 (음의 극성)



## V. PART DESIGN

### 4. Dress-Up Features

생성한 솔리드를 여러 가지 형태로 편집하는  
아이콘을 모아 놓은 툴 바



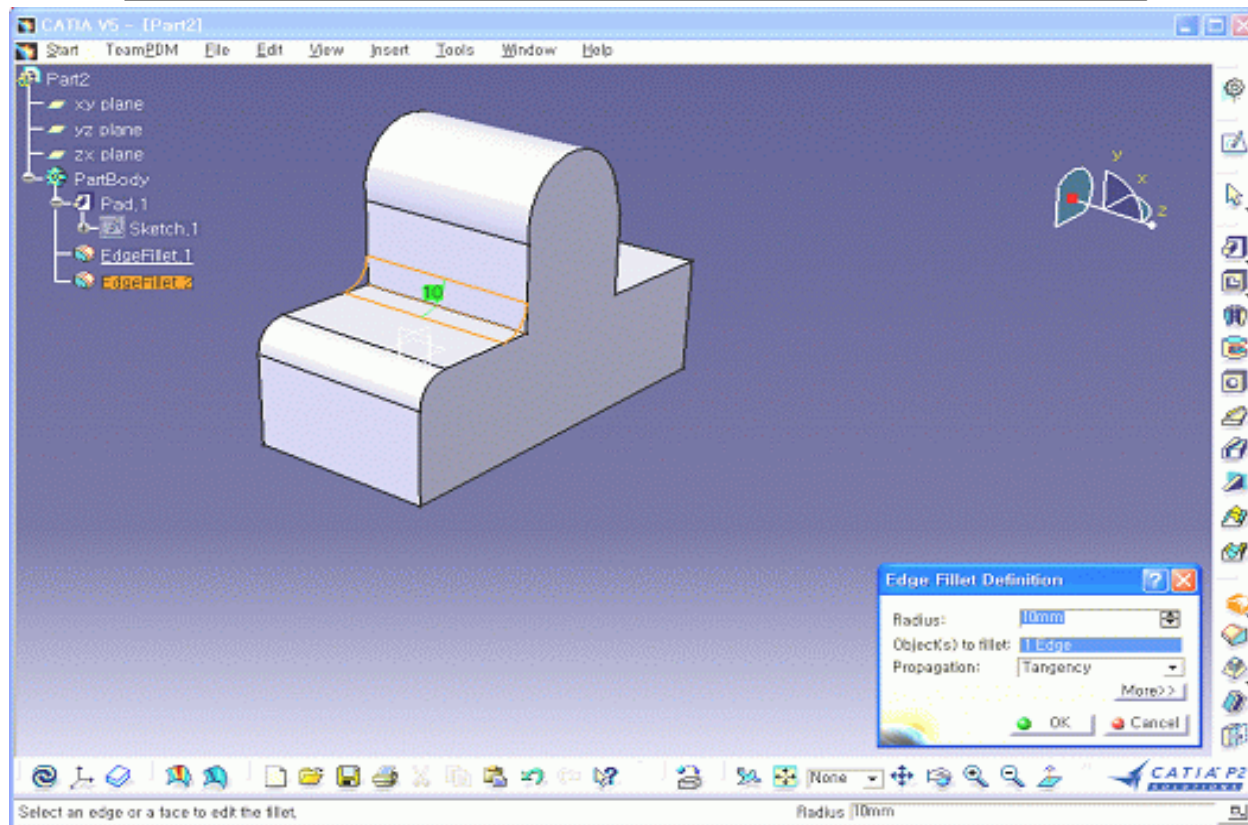
## V. PART DESIGN

### 4-1. Edge Fillet



EDGE FILLET

Edge를 선택하여 라운딩 작업을 수행



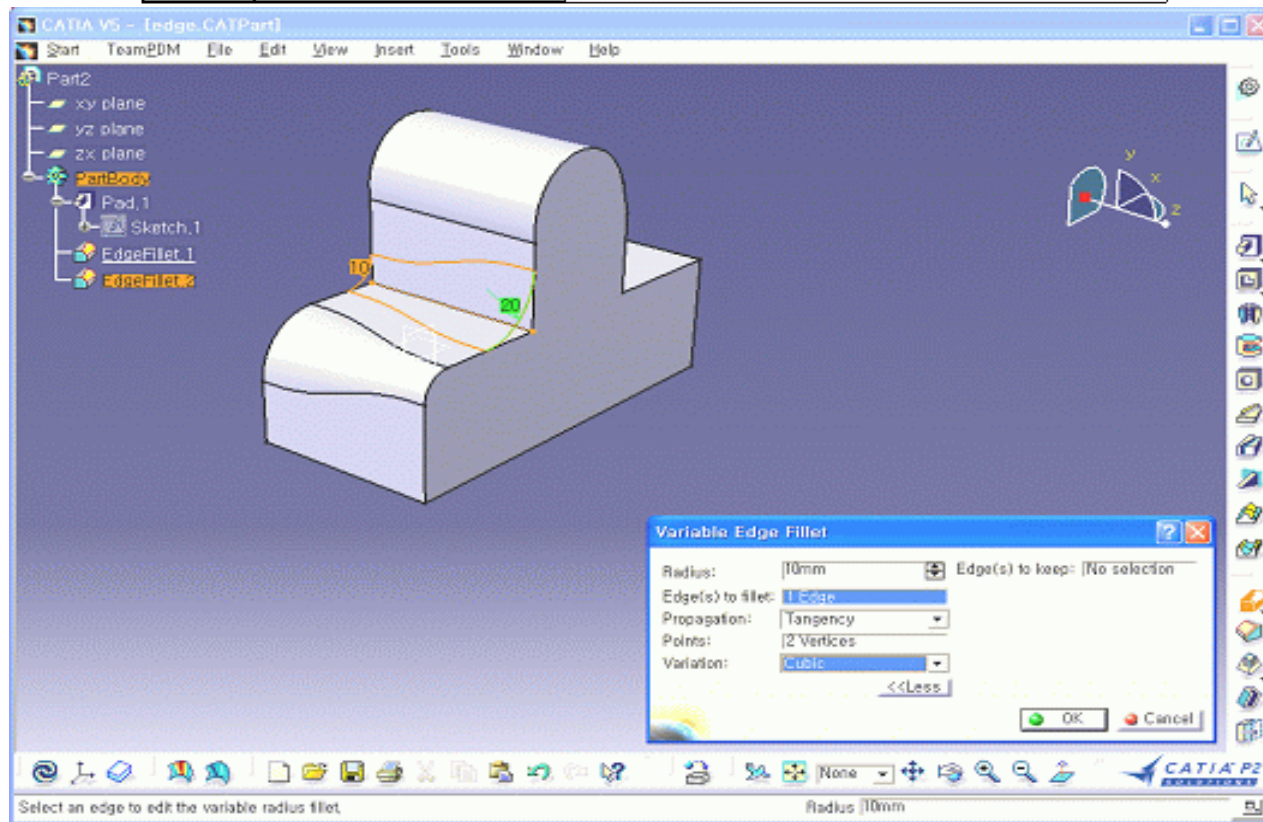
## V. PART DESIGN

### 4-2. Variable Radius Fillet



VARIABLE  
RADIUS FILLET

반경이 다른 라운딩 작업을 수행





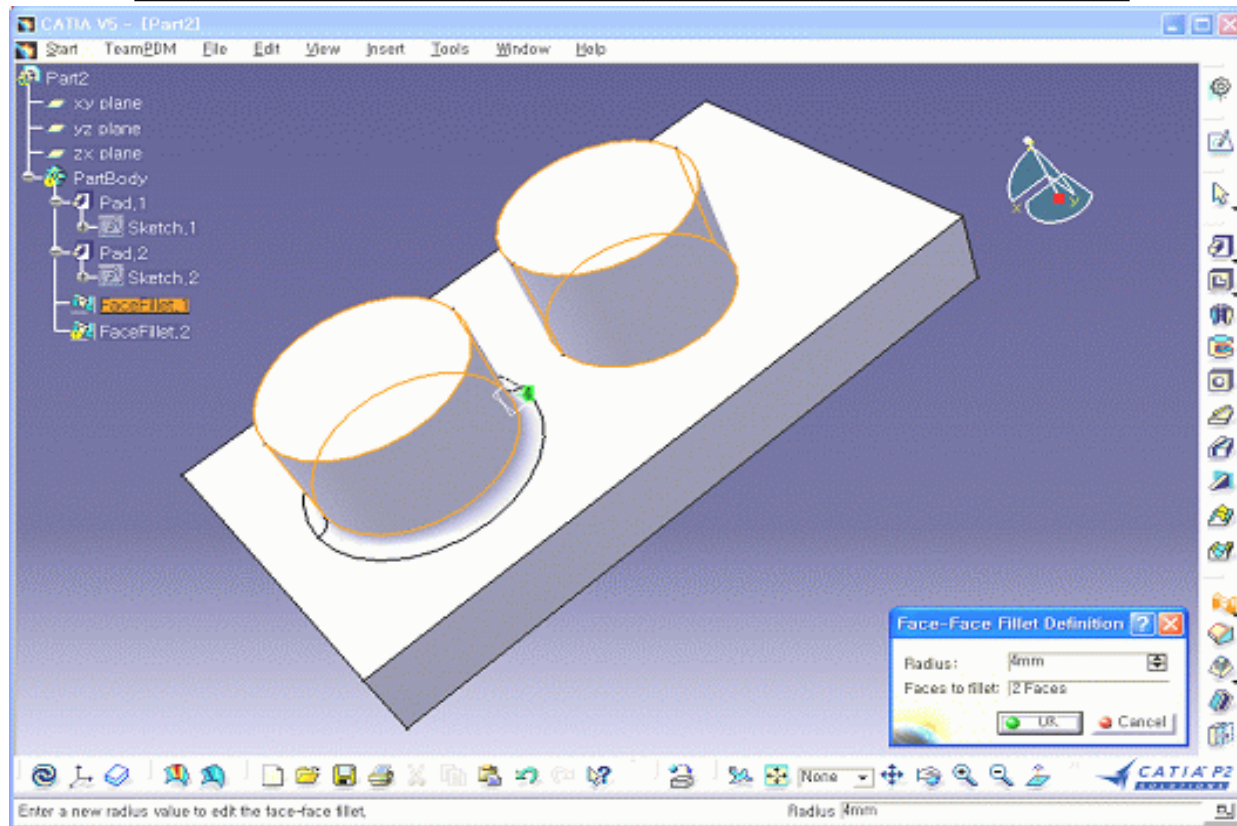
## V. PART DESIGN

### 4-3. Face-Face Fillet



FACE-FACE  
FILLET

Face 와 Face 사이에 라운딩 작업



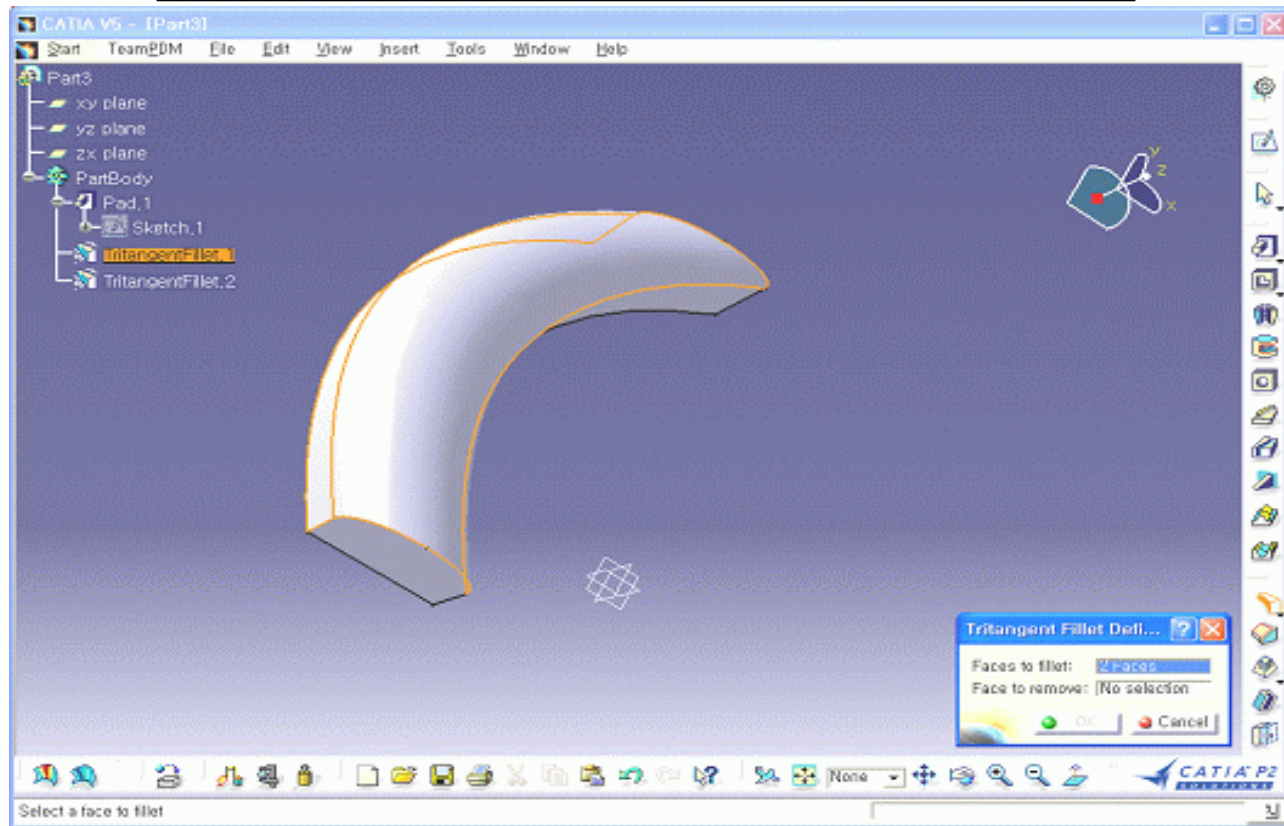
## V. PART DESIGN

### 4-4. Tritangent Fillet



**TRITANGENT  
FILLET**

두 면에 대한 Fillet 작업을 수행



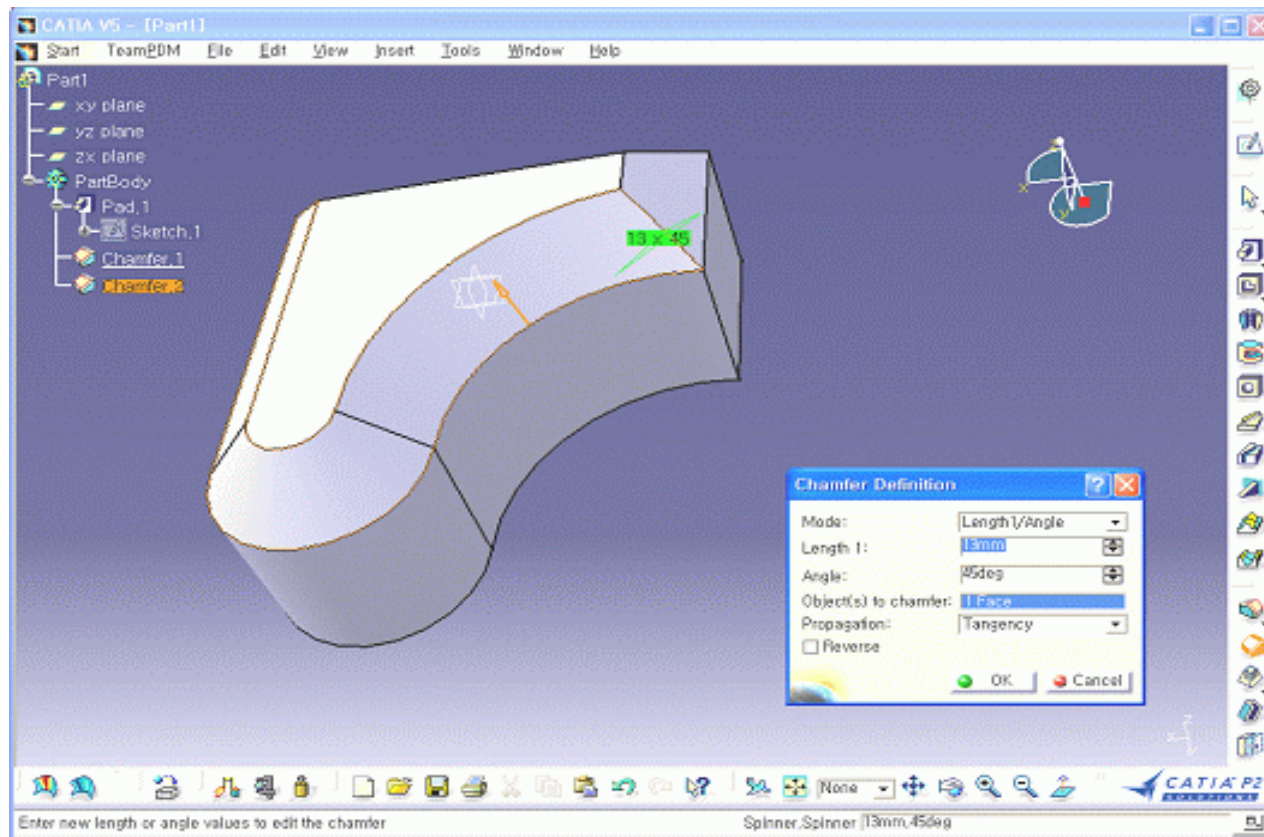
## V. PART DESIGN

### 4-5. Chamber



CHAMBER

모따기 작업을 수행



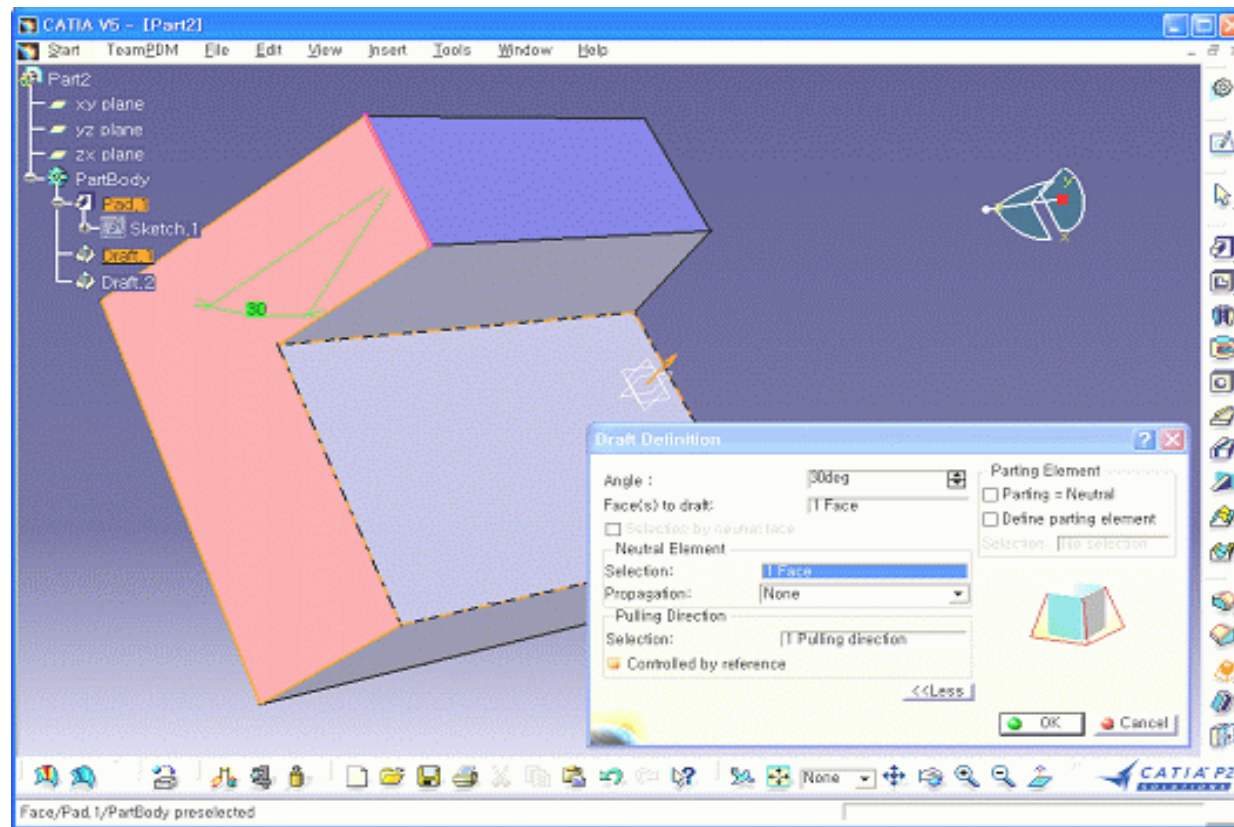
## V. PART DESIGN

### 4-6. Draft



DRAFT

제품형상에 Drafting 작업을 수행





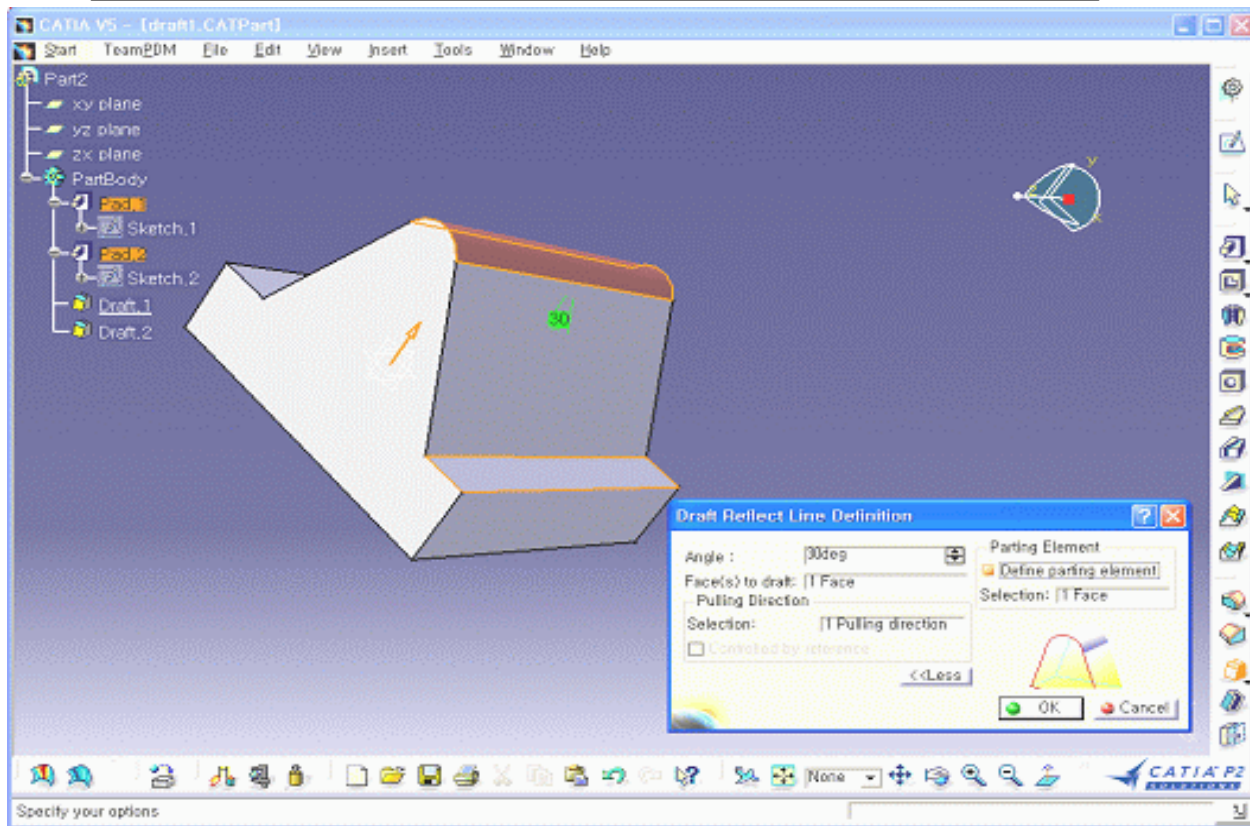
# V. PART DESIGN

## 4-7. Draft Reflect Line




**DRAFT  
REFLECT LINE**

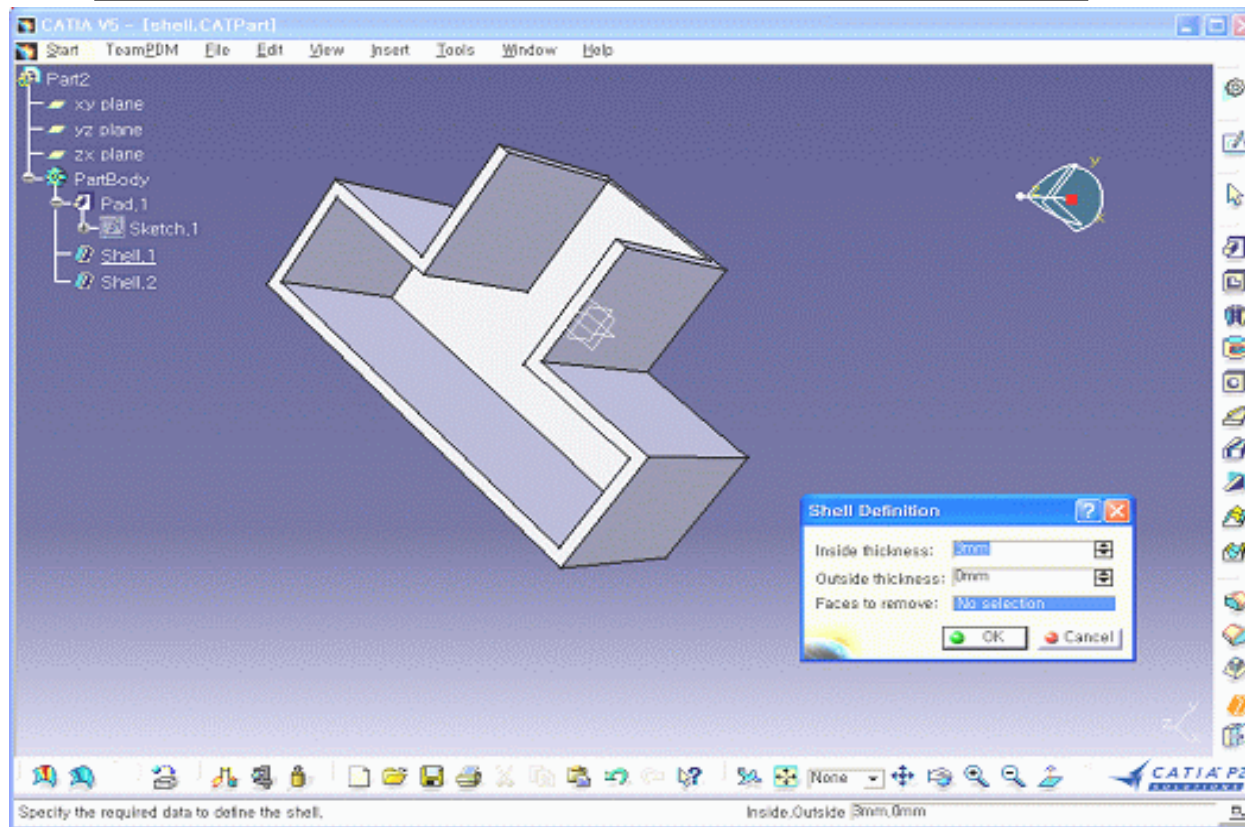
Reflect Line을 이용한 drafting을 수행



# V. PART DESIGN

## 4-8. Shell

	SHELL	제품의 형상에서 일정한 부분을 빼는 작업
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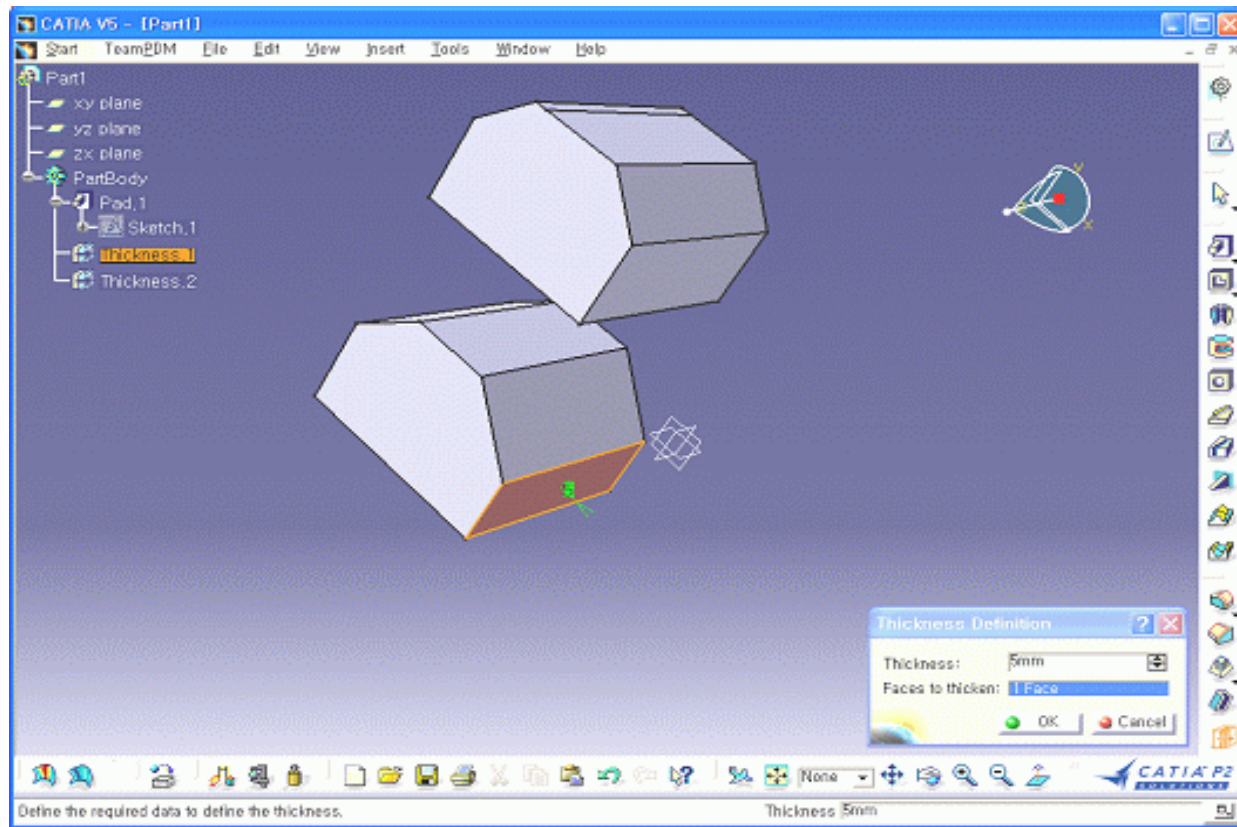
## V. PART DESIGN

### 4-9. Thickness



#### THICKNESS

제품형상에 사용자가 두께를 늘리고  
자 할때 사용하는 기능



## V. PART DESIGN

### 5. Surface-Based Features

Wire-frame and Surface 에서 생성한 Surface를  
솔리드 화 하거나 편집하는 기능을 모아 놓은 툴바





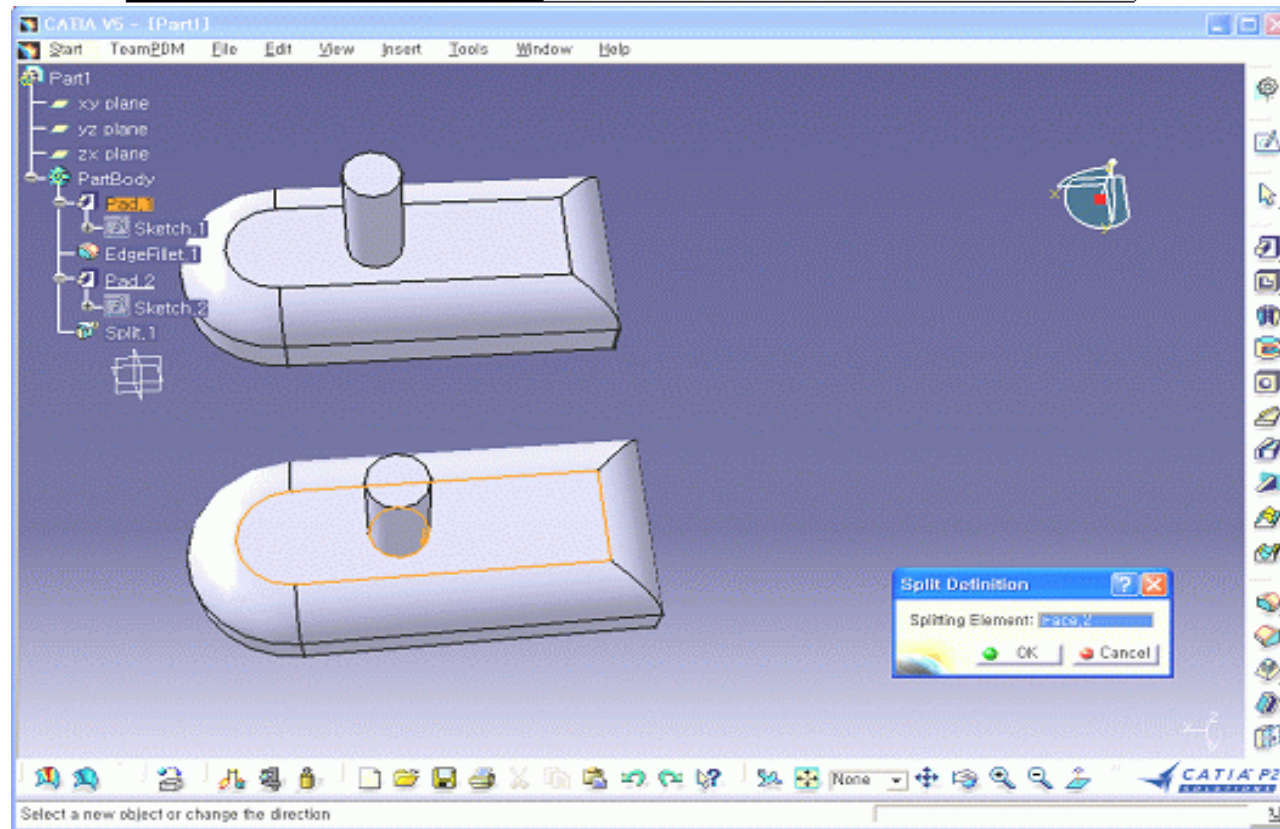
# V. PART DESIGN

## 5-1. Split



SPLIT

형상의 일부분을 임의 지점까지 잘라낼 때 사용하는 기능



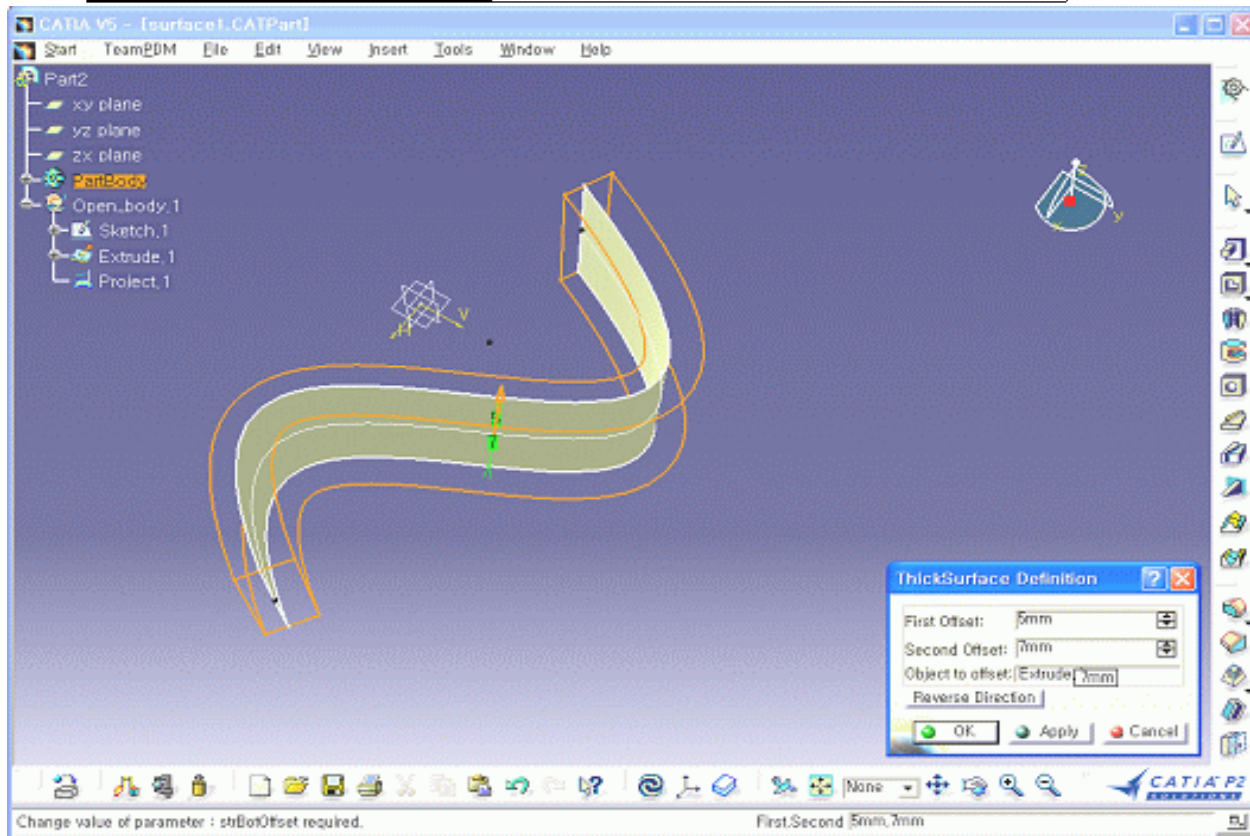
# V. PART DESIGN

## 5-2. Thick Surface



### THICK SURFACE

실제 surface를 이용하여 그 surface  
에 두께를 부여하면서 Solid 생성



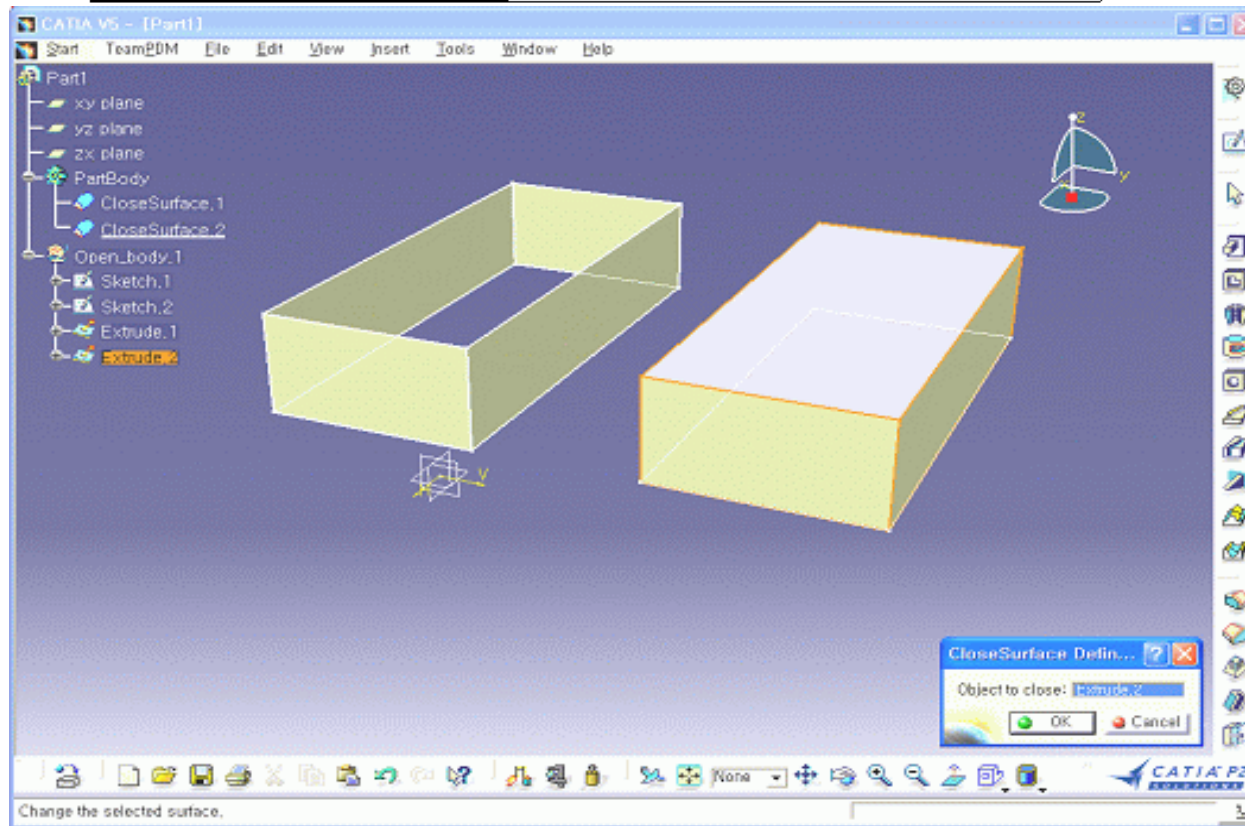
## V. PART DESIGN

### 5-3. Close Surface



**CLOSE  
SURFACE**

Surface를 솔리드로 채워주는 기능



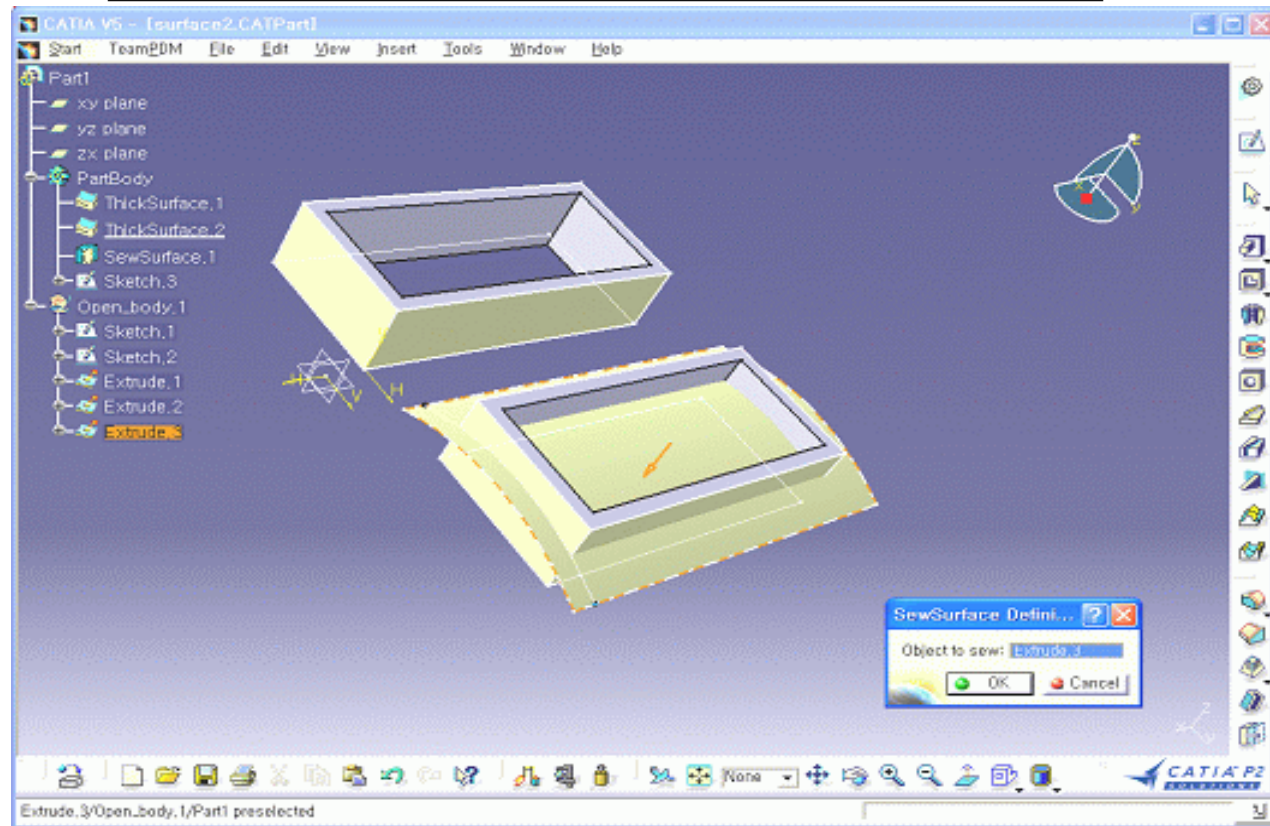
## V. PART DESIGN

### 5-4. Sew Surface



SEW SURFACE

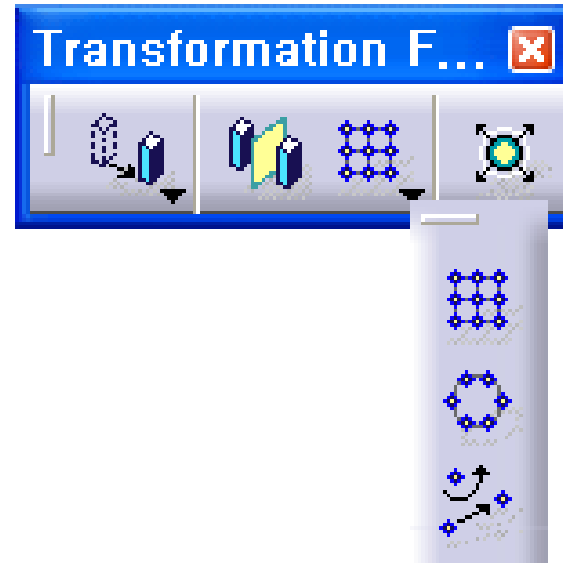
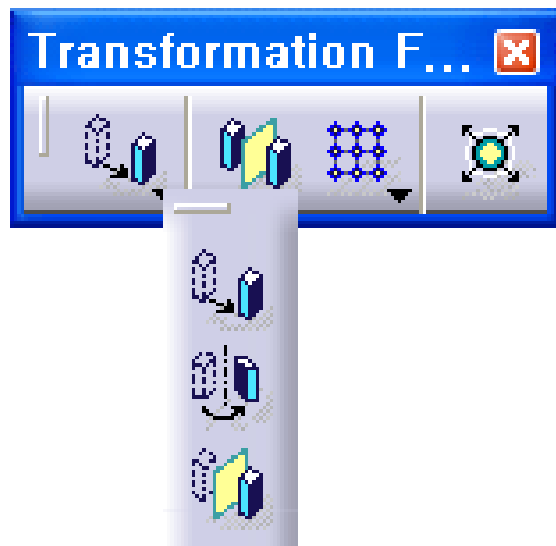
Surface를 솔리드에 붙여는 기능



## V. PART DESIGN

### 6. Transformation Features


솔리드의 이동이나 복사등의 아이콘을 모아놓은 툴바로 Feature 들을 변형 할 때 사용하는 기능들이 조합

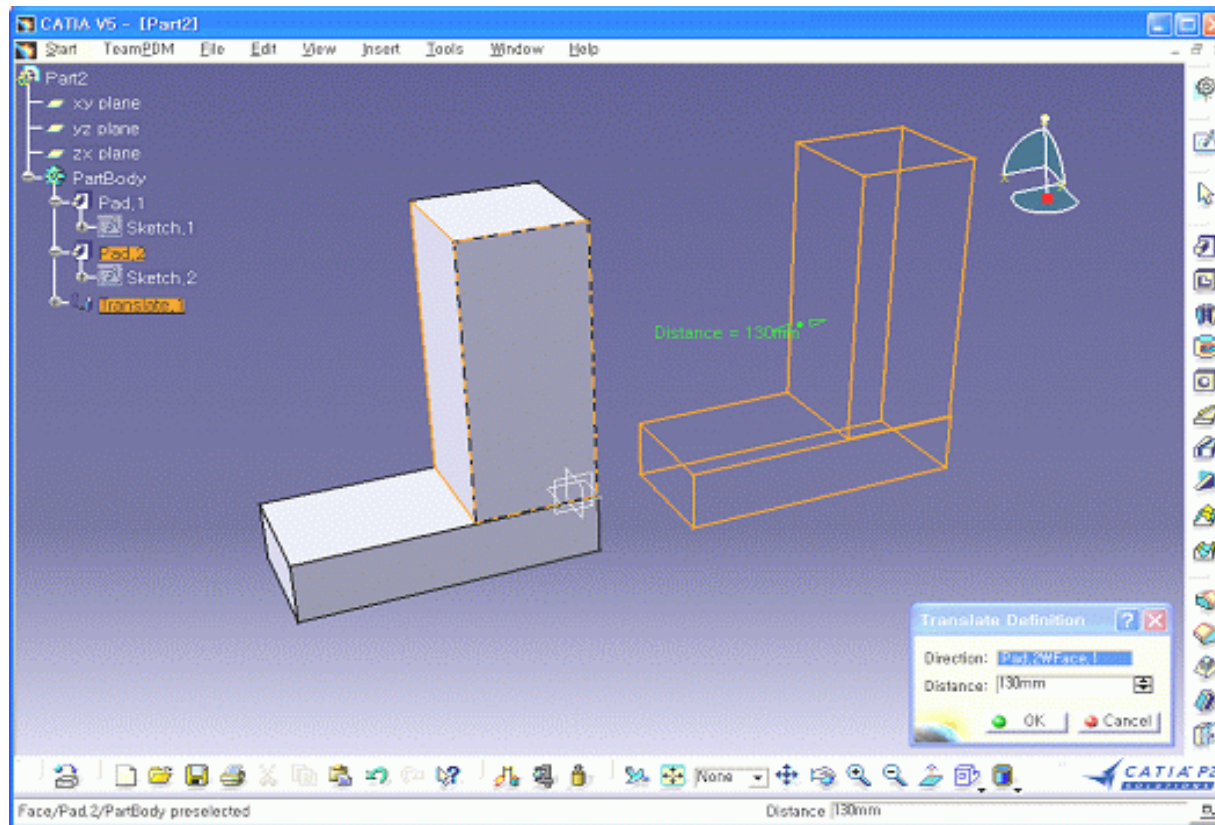




## V. PART DESIGN

### 6-1. Translation

	<b>TRANSLATION</b>	생성되어 있는 Solid 자체를 이동
---	--------------------	----------------------



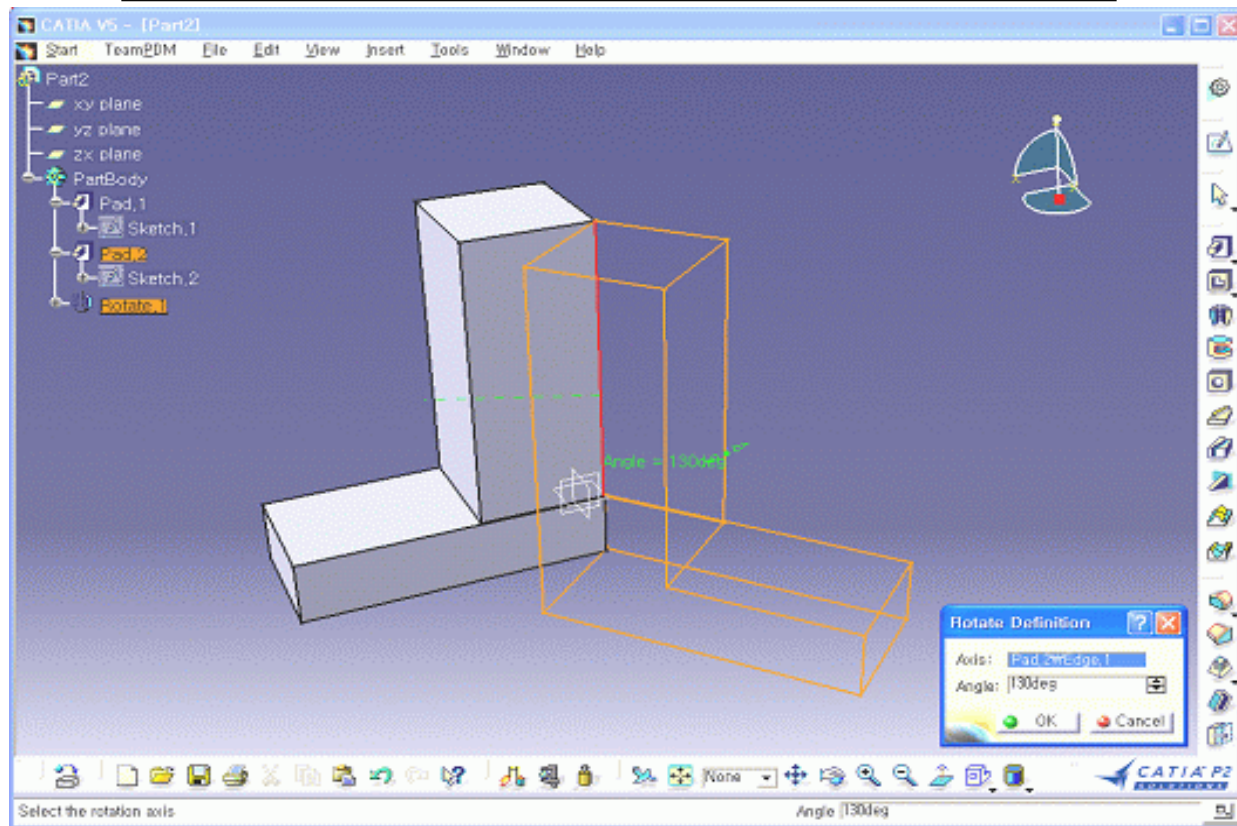
## V. PART DESIGN

### 6-2. Rotation



#### ROTATION

생성되어 있는 Solid 자체를 회전





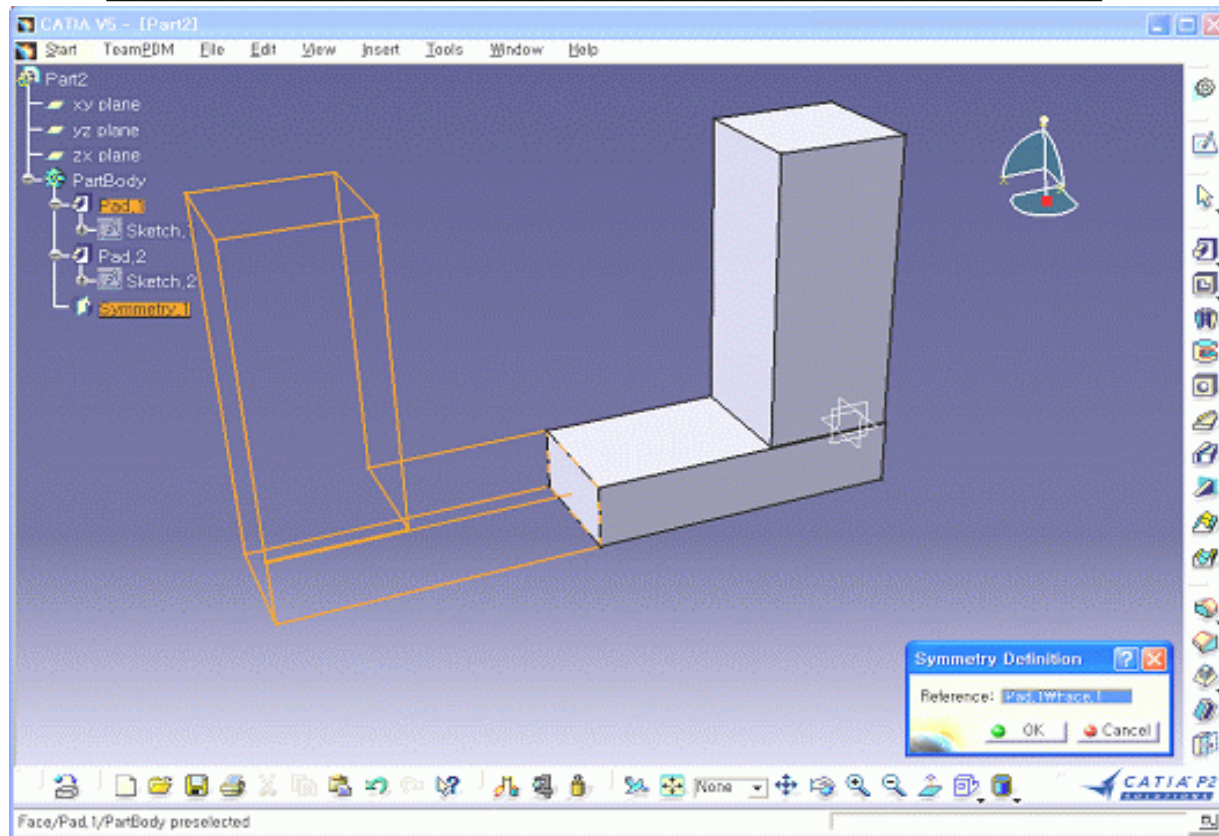
## V. PART DESIGN

### 6-3. Symmetry



SYMMETRY

생성되어 있는 Solid 자체를 대칭이동



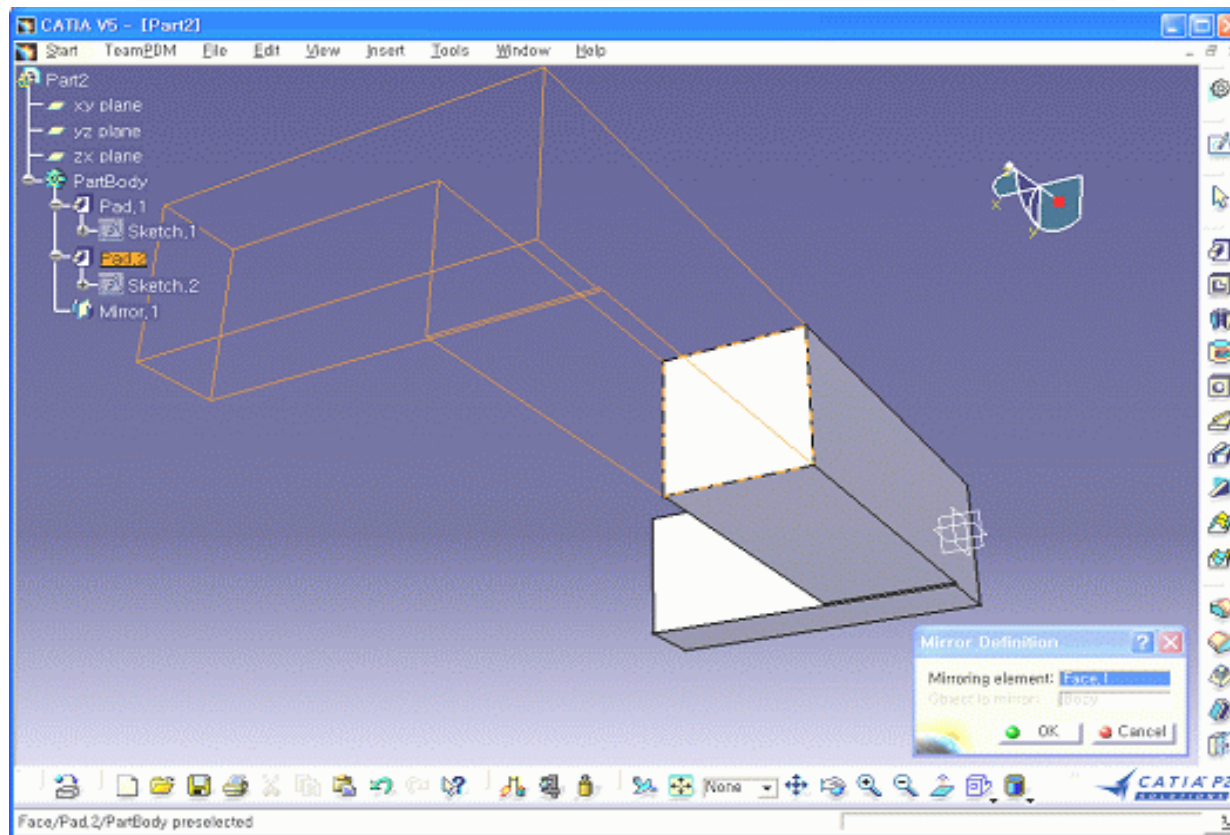
# V. PART DESIGN

## 6-4. Mirror




### MIRROR

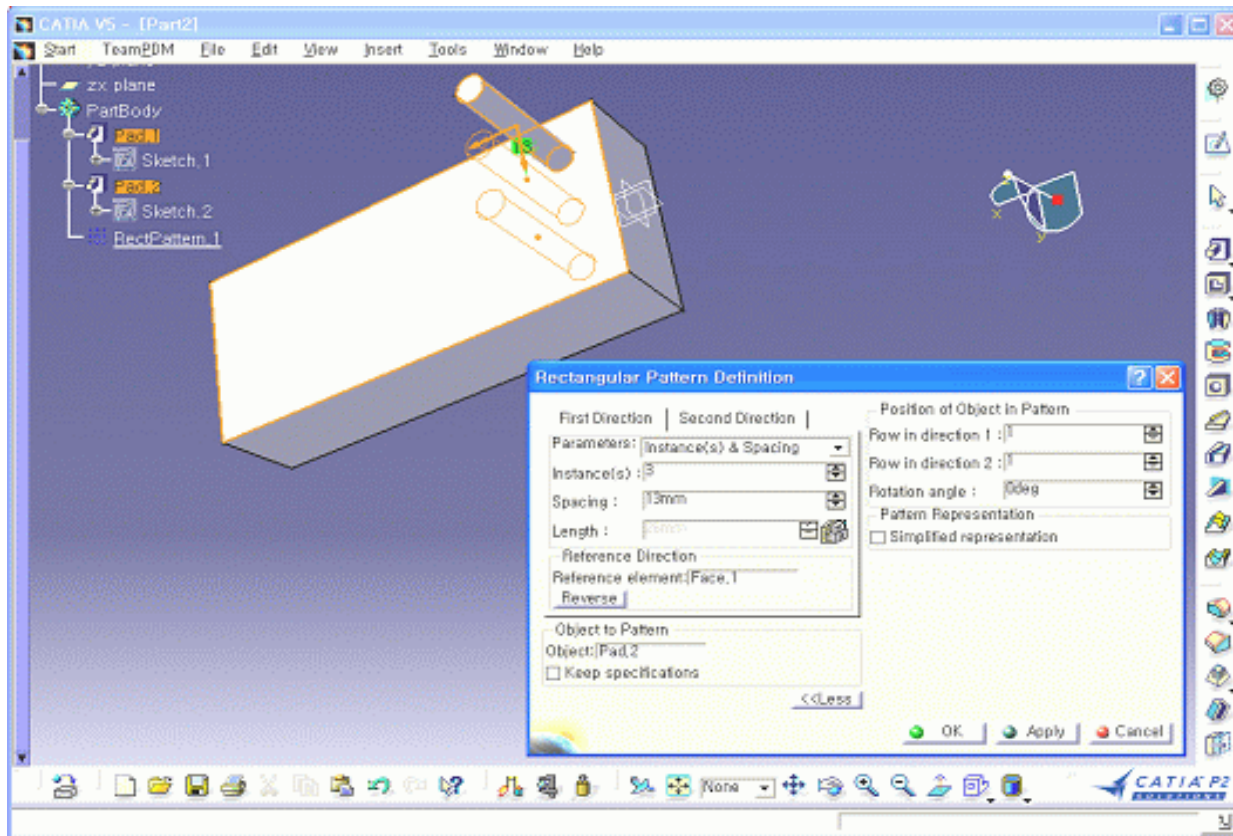
똑같은 형상을 반대편에도 같은 모양으로 만들 때 사용



# V. PART DESIGN

## 6-5. Rectangular Pattern

	<b>RECTANGULAR PATTERN</b>	바둑판 모양으로 여러개의 body를 copy 형태로 같은 모양으로 생성
---	--------------------------------	--



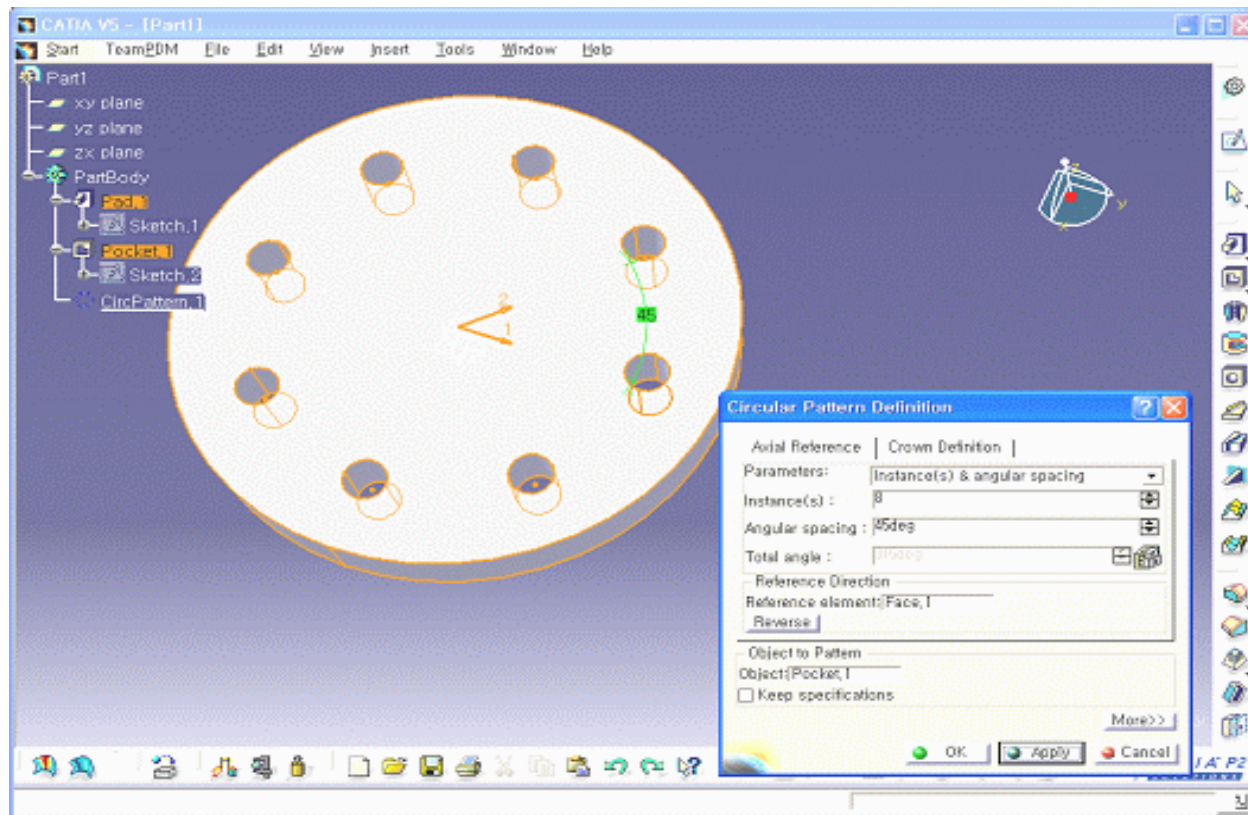
# V. PART DESIGN

## 6-6. Circular Pattern



**CIRCULAR  
PATTERN**

원형 모양으로 여러개의 body를 copy  
형태로 같은 모양으로 생성





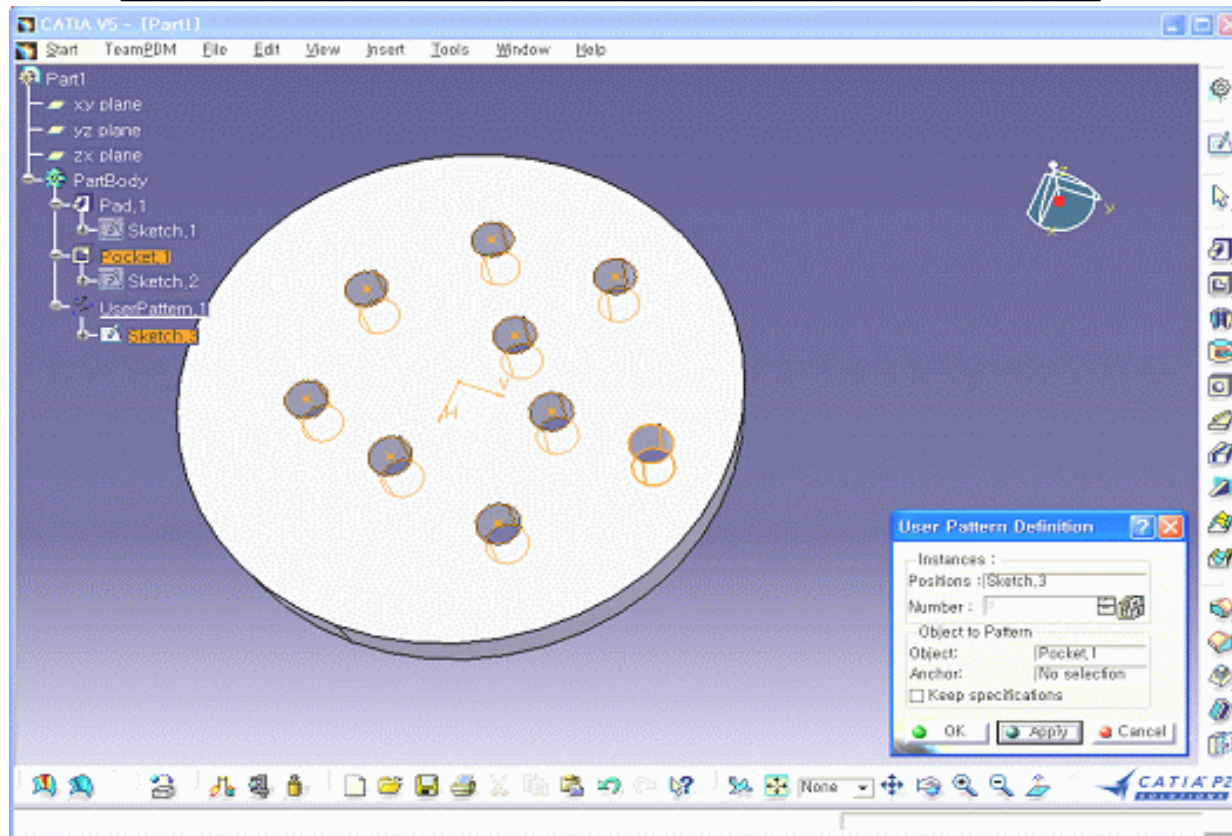
## V. PART DESIGN

### 6-7. User Pattern



USER PATTERN

사용자가 정의한 형태로 Pattern 처리



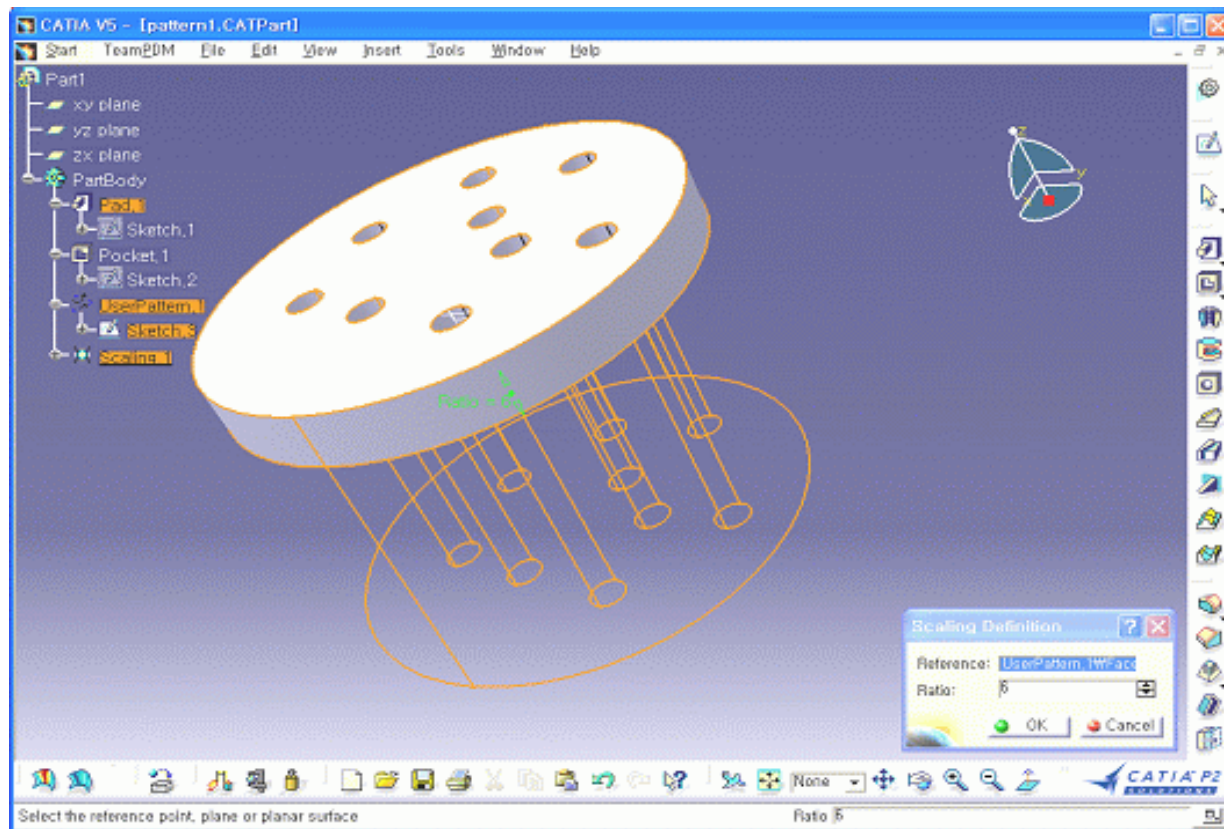
## V. PART DESIGN

### 6-8. Scaling



SCALING

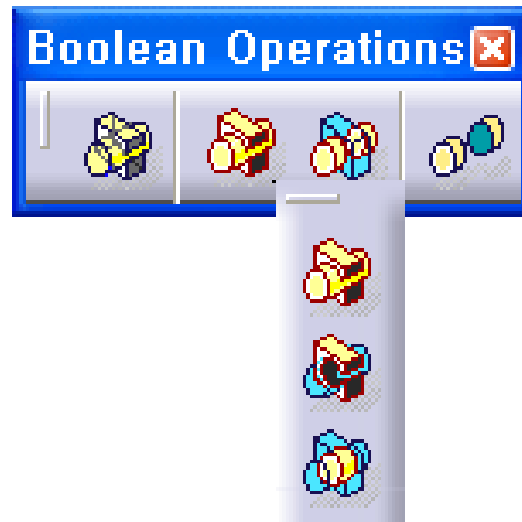
형상의 크기를 조절하는 기능



## V. PART DESIGN

### 7. Boolean Operations

Body 와 Body를 Boolean 연산을 해주는 아이콘을 모아놓은 툴 바





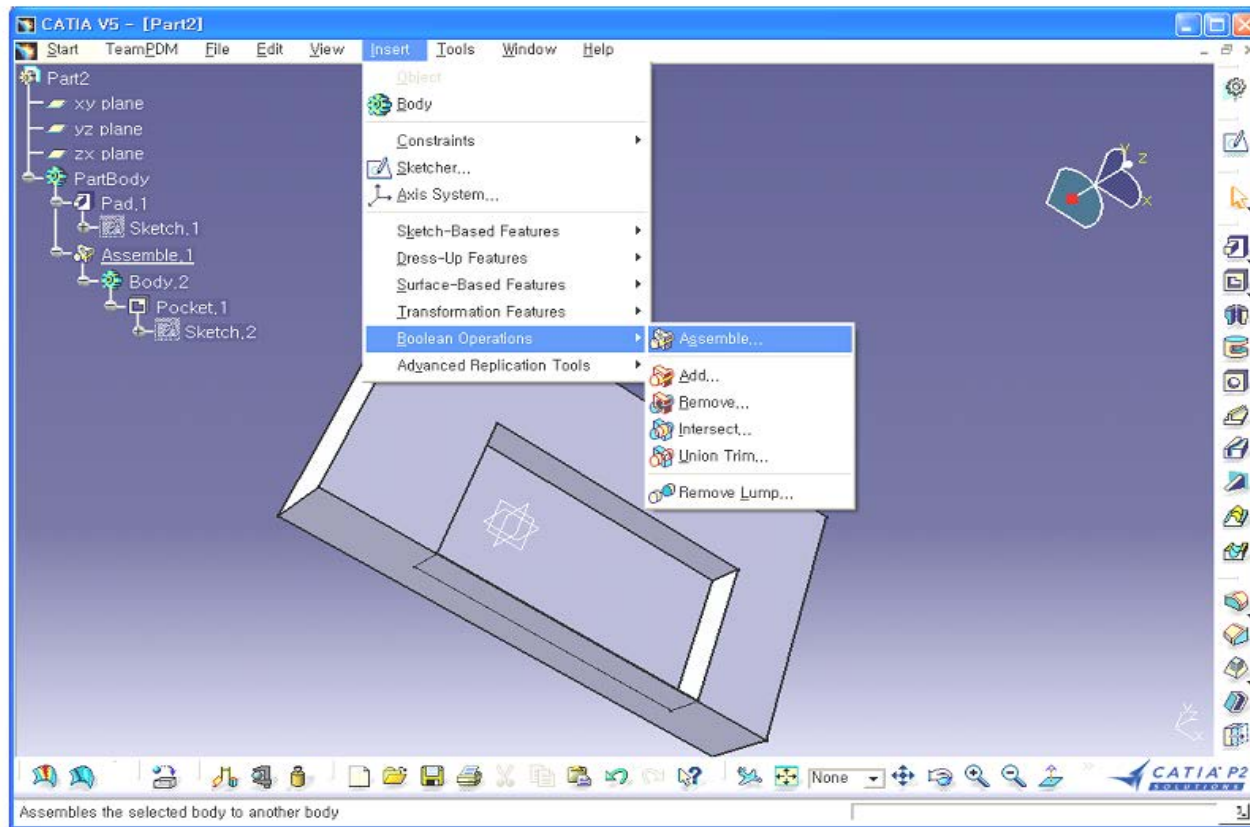
# V. PART DESIGN

## 7-1. Assemble




ASSEMBLE

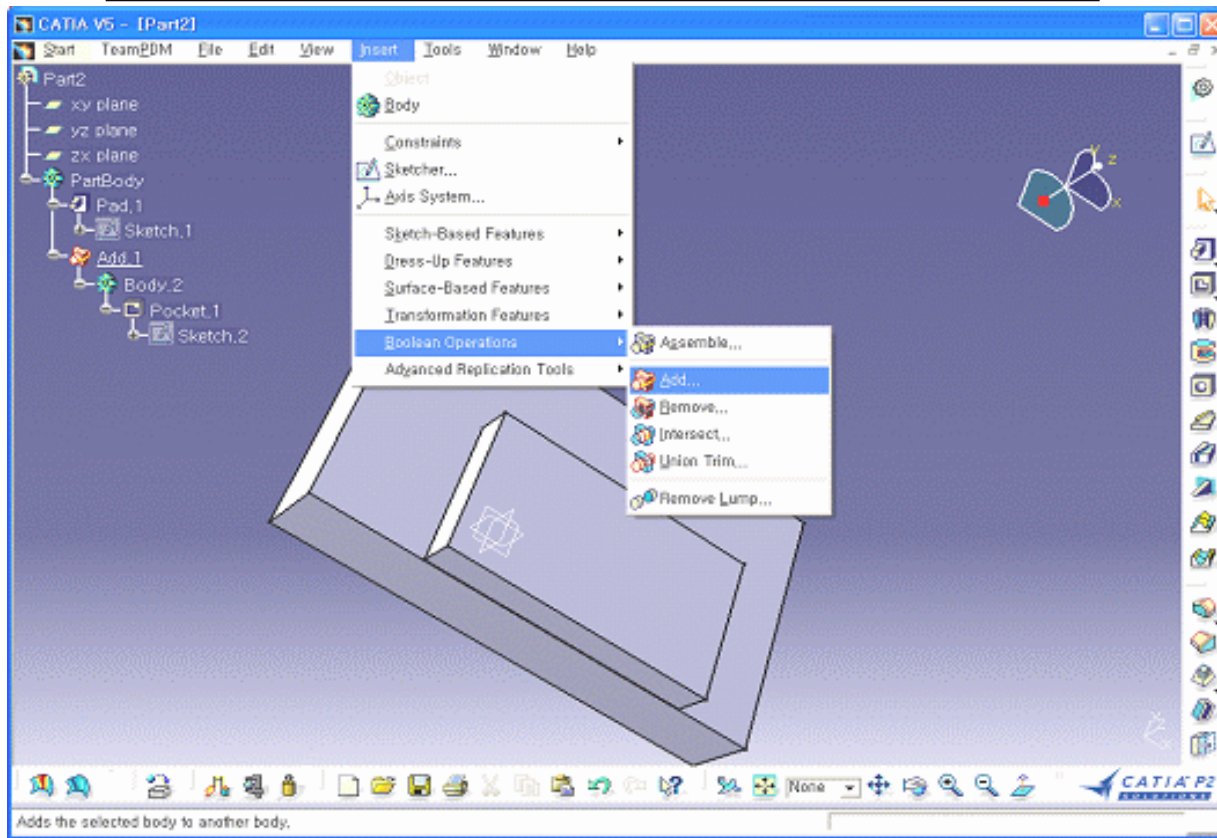
극성을 유지하면서 Union 작업을 수행



# V. PART DESIGN

## 7-2. Add

	ADD	+극성을 유지하면서 Union 작업을 수행
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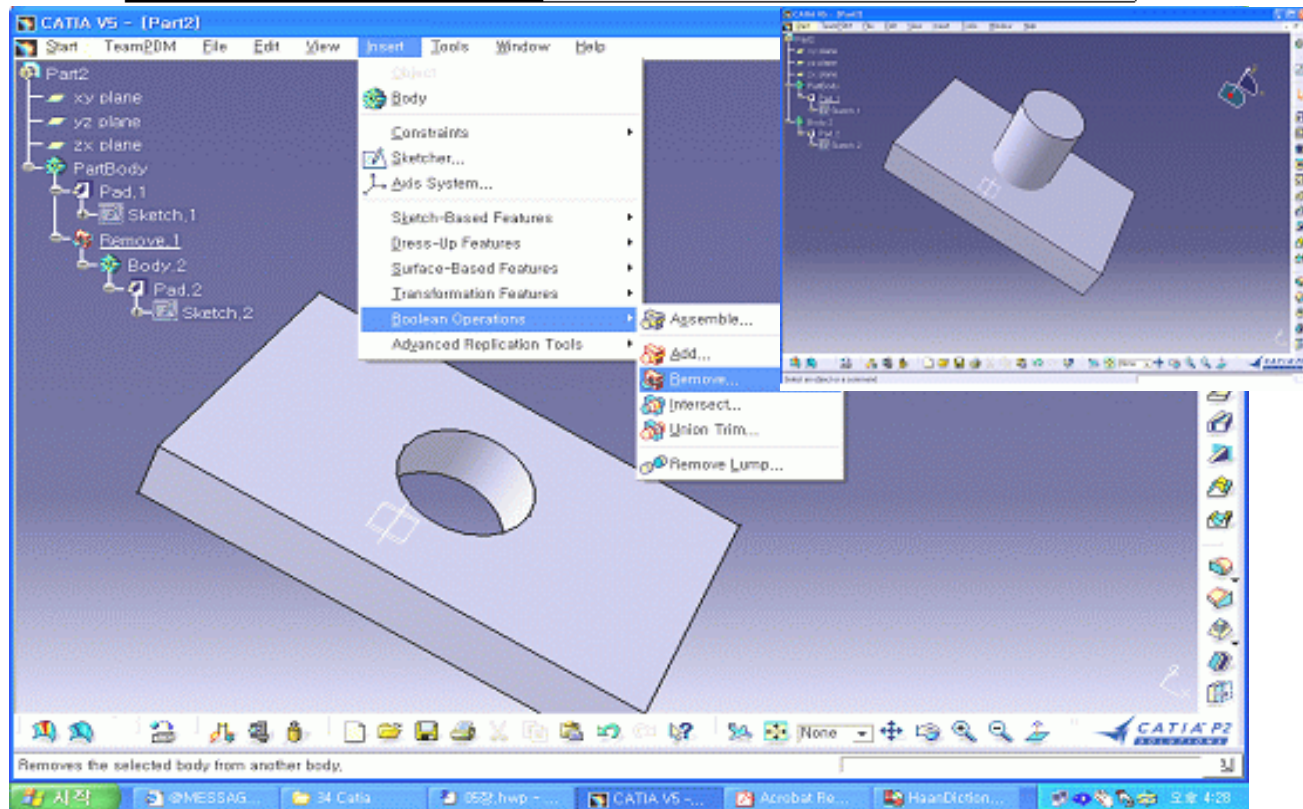
## V. PART DESIGN

### 7-3. Remove



REMOVE

Subtract 작업을 수행



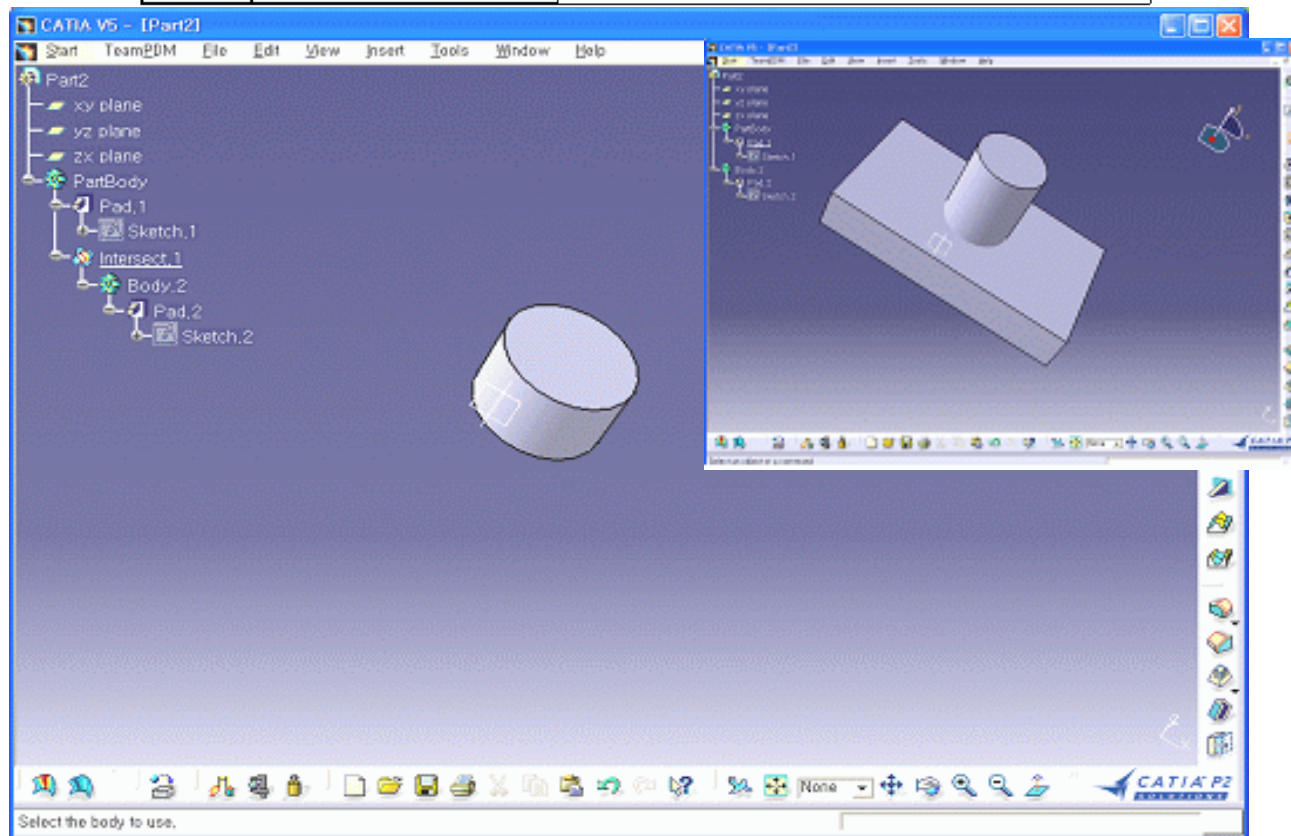
## V. PART DESIGN

### 7-4. Intersect



INTERSECT

서로 교차되는 부분만을 남겨 생성





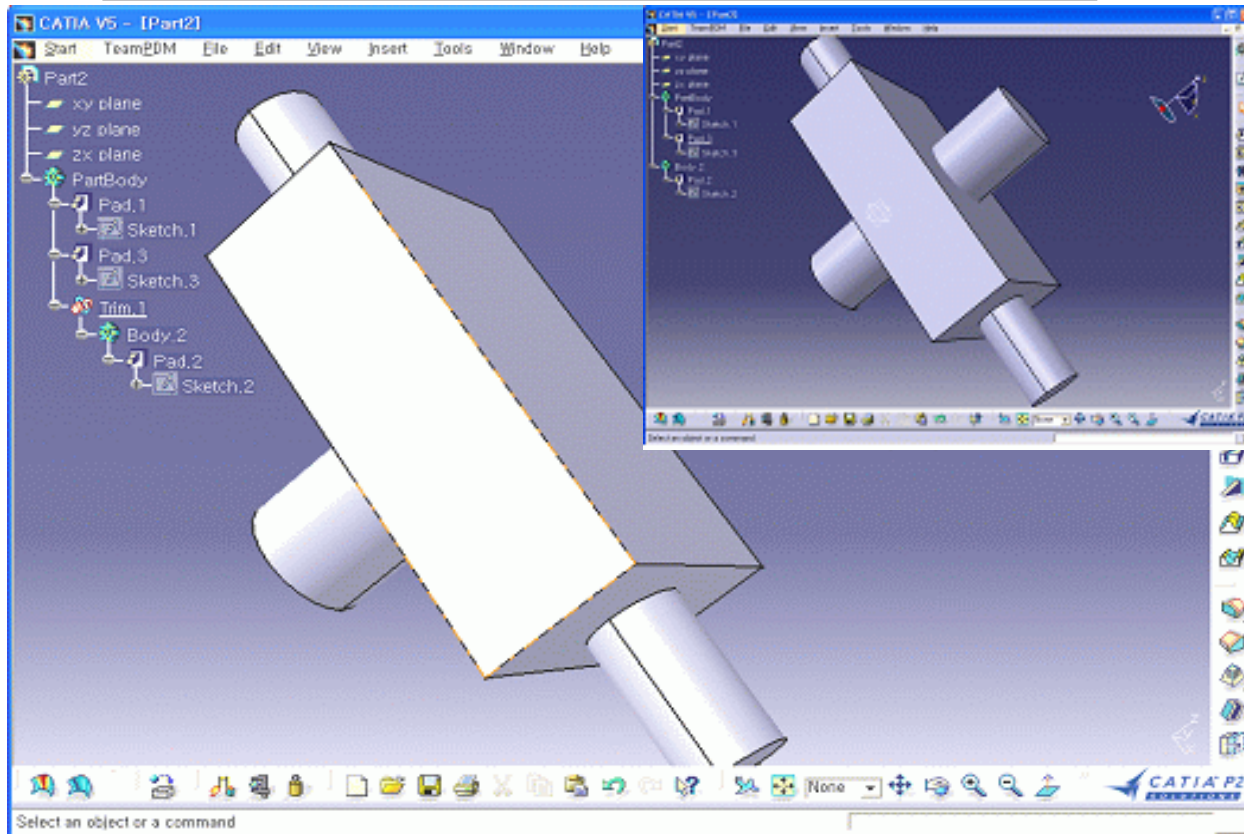
## V. PART DESIGN

### 7-5. Union Trim



UNION TRIM

Union 작업시 trim작업을 동시에 수행



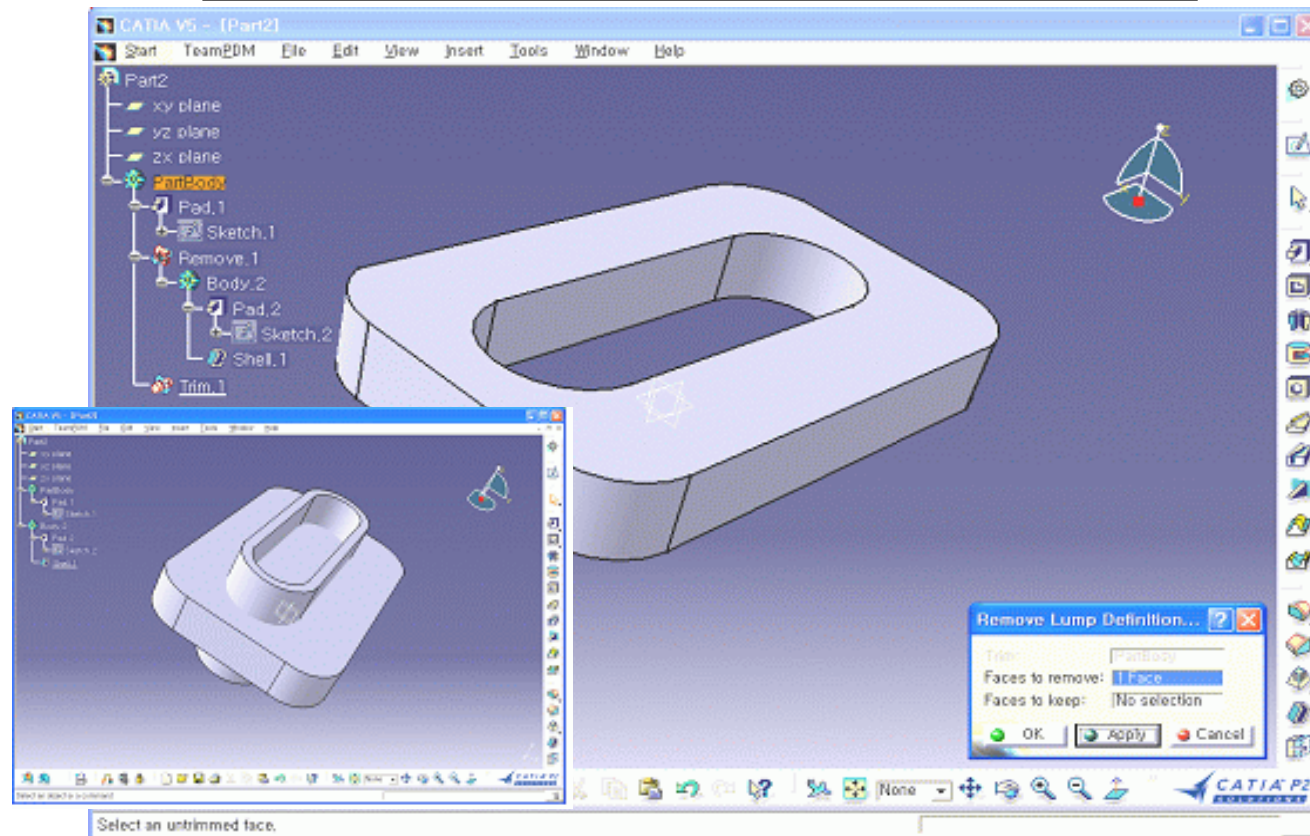
# V. PART DESIGN

## 7-6. Remove Lump



REMOVE LUMP

Remove 작업시 trim 작업을 동시 수행




## V. PART DESIGN


### 8. Reference Element




#### 8-1. Point

	POINT	3 차원 공간상에 Point를 생성
---	-------	---------------------

#### 8-2. Line

	LINE	3 차원 공간상에 Line 을 생성
---	------	---------------------

#### 8-3. Plane


	PLANE	3 차원 공간상에 임의의 평면을 생성
---	-------	----------------------




## V. PART DESIGN

### 9. Measure


#### 9-1. Measure Between

	MEASURE BETWEEN	물체간의 치수를 검사하는 기능
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#### 9-2. Measure

	MEASURE	Edge (line/curve/arc) 치수를 검사
---	---------	------------------------------


#### 8-3. Measure Inertia

	MEASURE INERTIA	Part 의 Volume, Mass, Density을 확인
---	--------------------	----------------------------------


## V. PART DESIGN

### 10. Tools


#### 10-1. Update All

	UPDATE ALL	잘못된 수행을 수정
---	------------	------------

#### 10-2. Axis System

	AXIS SYSTEM	기준좌표축의 이동
---	-------------	-----------

#### 10-3. Open Catalog

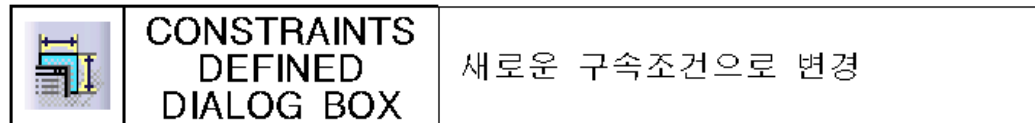
	OPEN CATALOG	카탈로그 열기
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## V. PART DESIGN

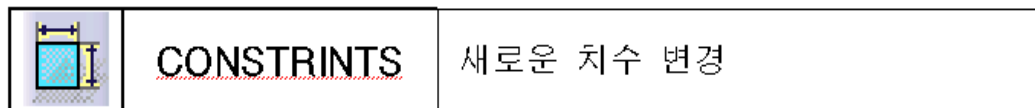
### 11. Constraints

3 차원 에 있는 솔리드와 Plane 사이의 구속조건을 변경  
하거나 솔리드의 Element의 치수를 나타내 준다.

#### 11-1. Constraints Defined Dialog Box




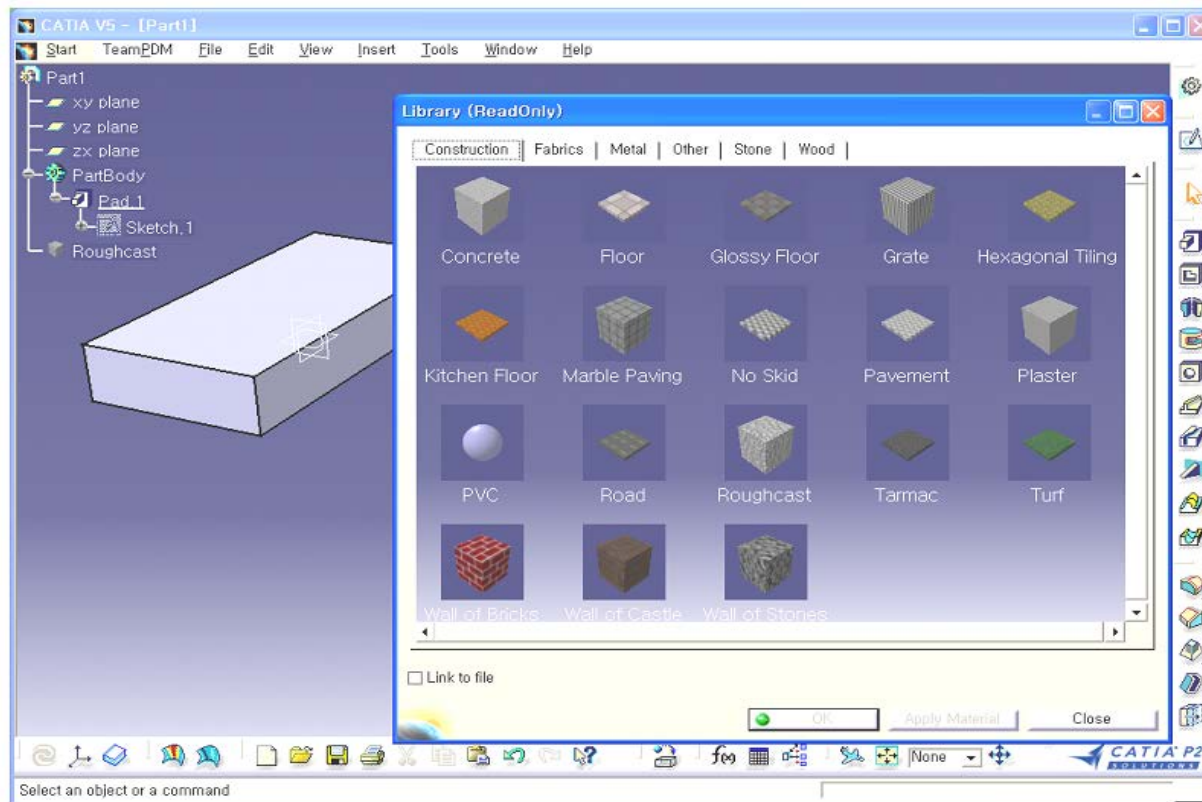
#### 11-2. Constraints



## V. PART DESIGN

### 12. Apply Material

	<b>APPLY MATERIAL</b>	솔리드 입체 재질 표현
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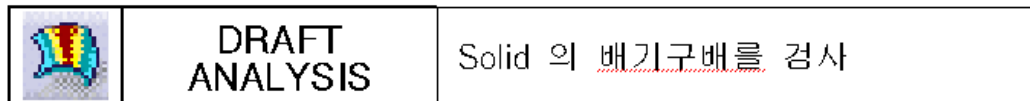


## V. PART DESIGN

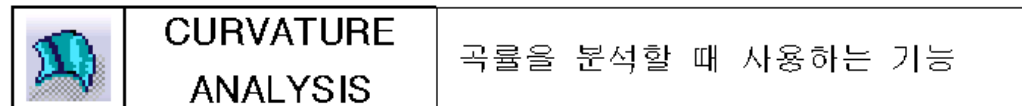
### 12. Apply Material

Draft 로 생성한 솔리드의 Draft를 분석

#### 12-1. Draft Analysis

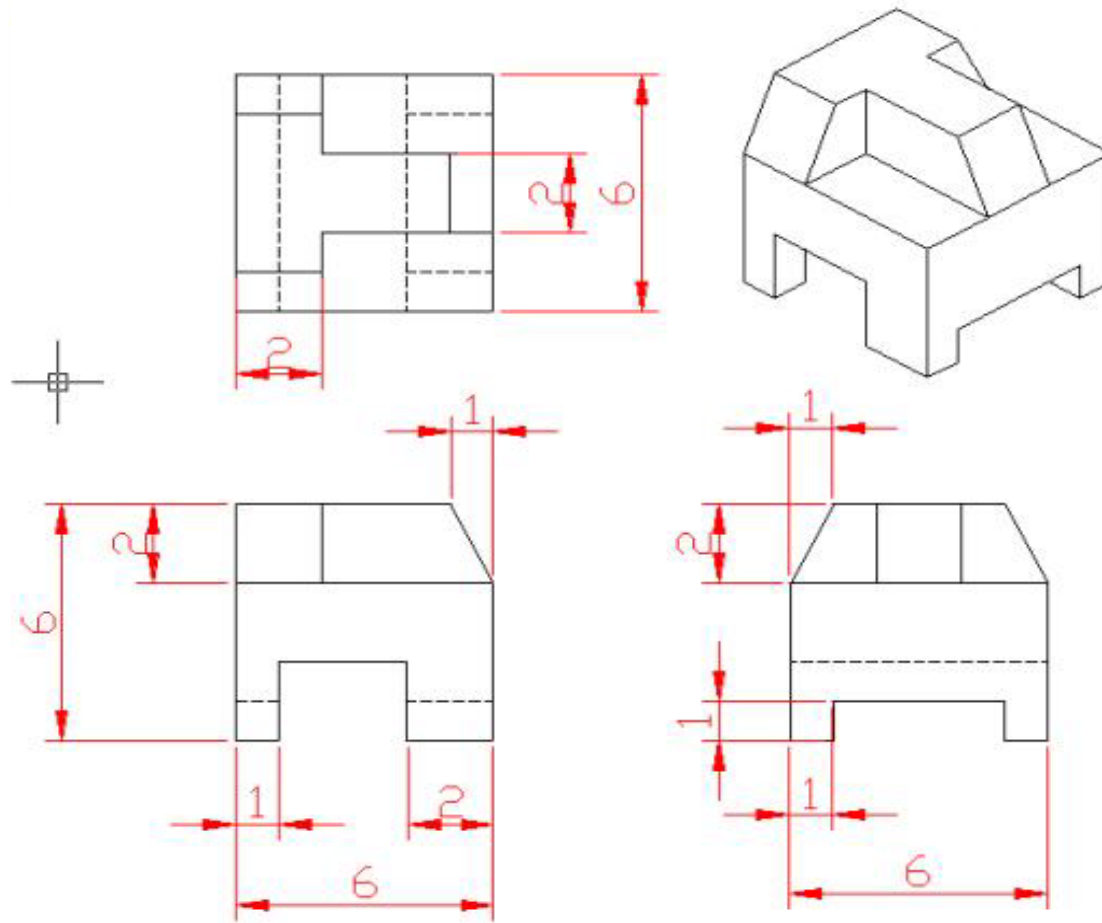


#### 12-2. Curvature Analysis



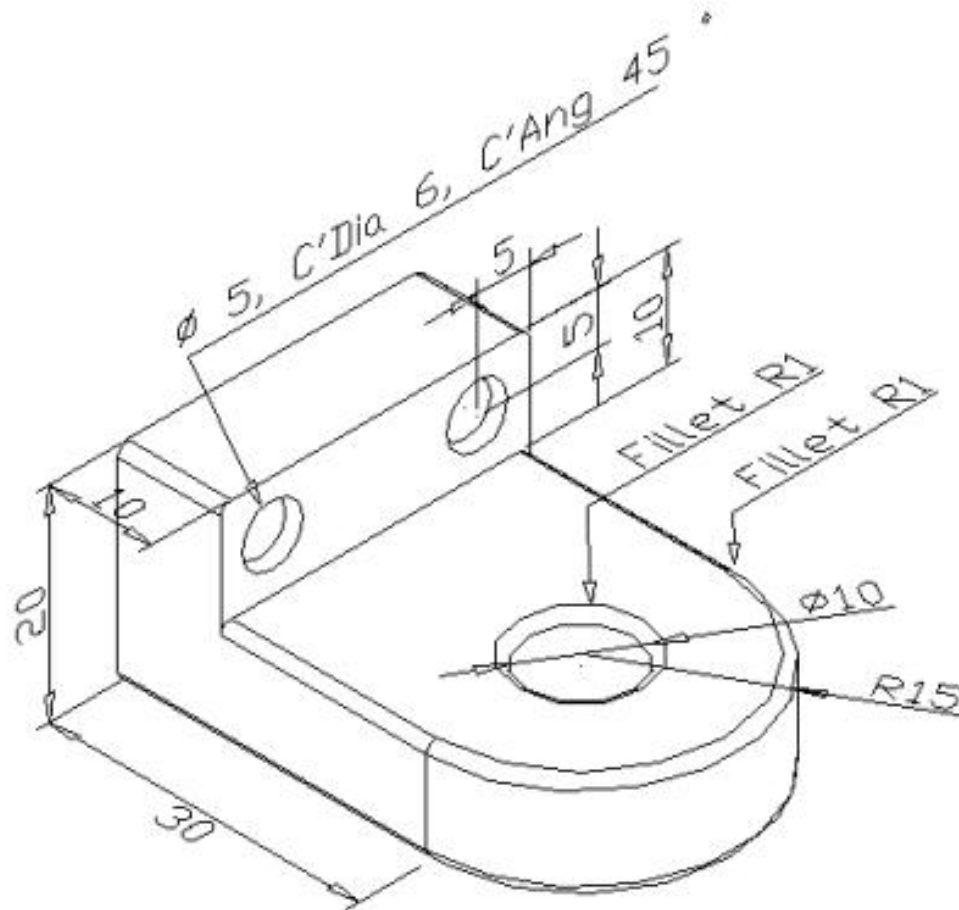
## V. PART DESIGN

### 13. 예 제 도 면 (1)



## V. PART DESIGN

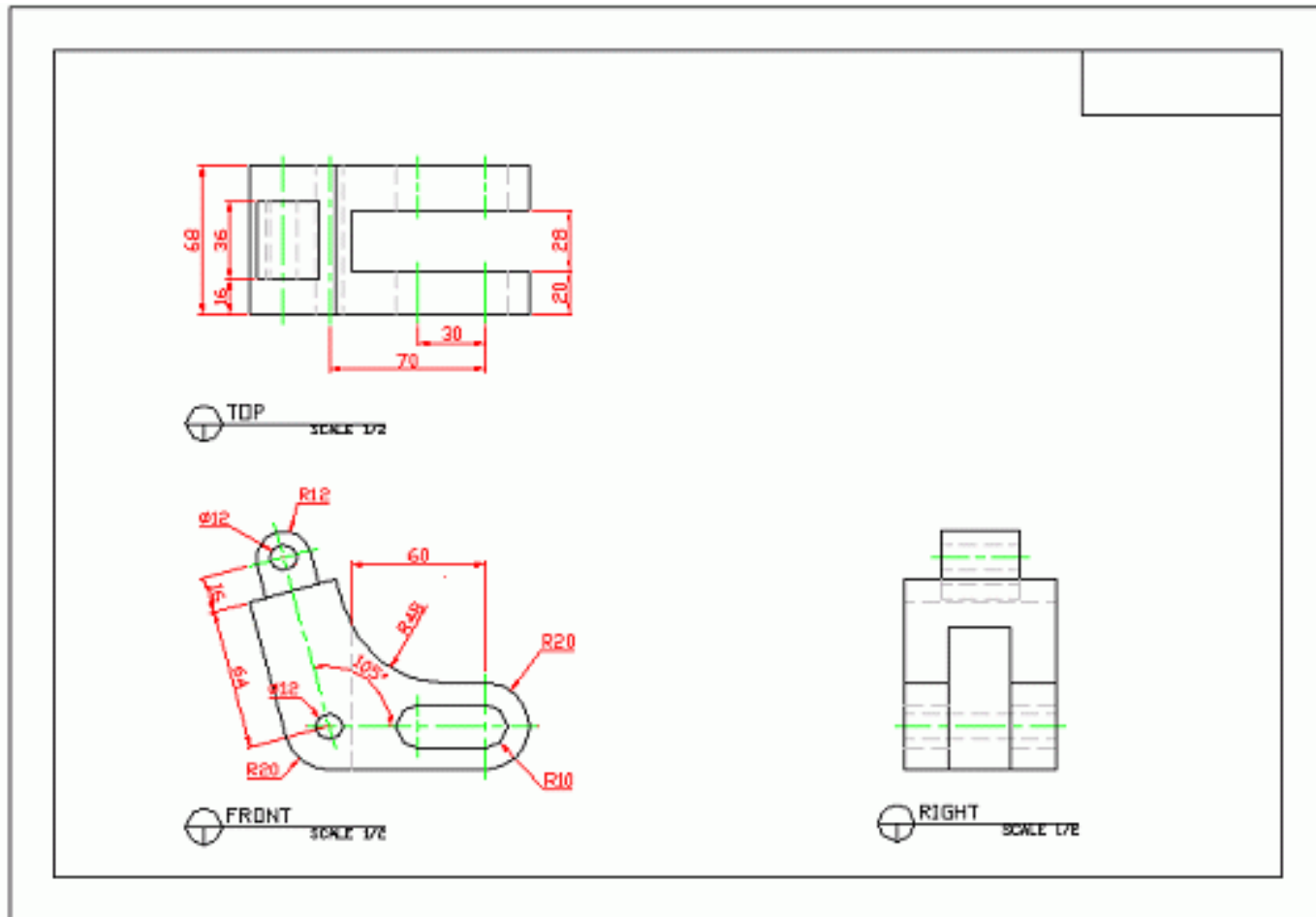
### 13. 예 제 도 면 (2)





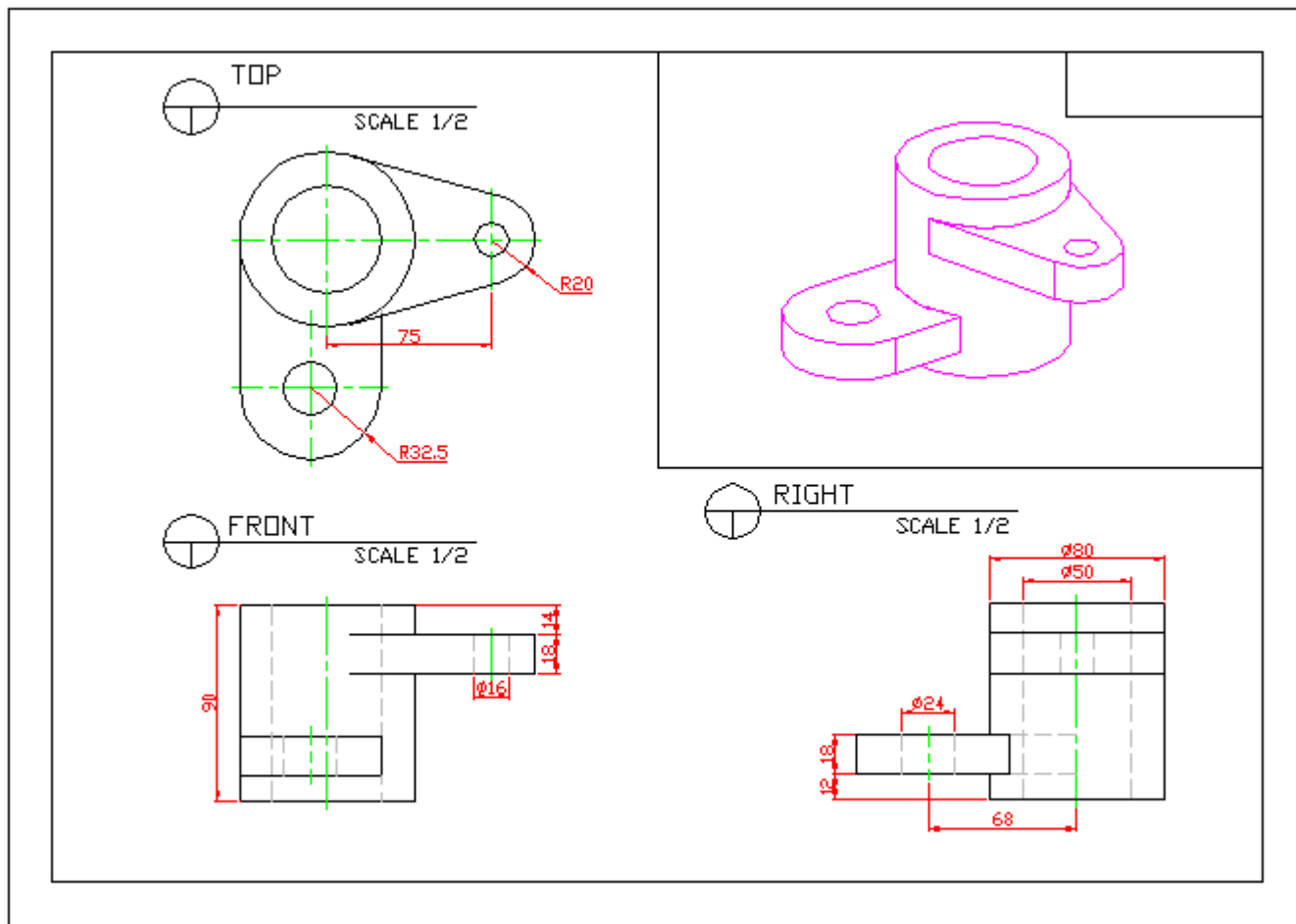
## V. PART DESIGN

### 13. 예 제도면 (3)



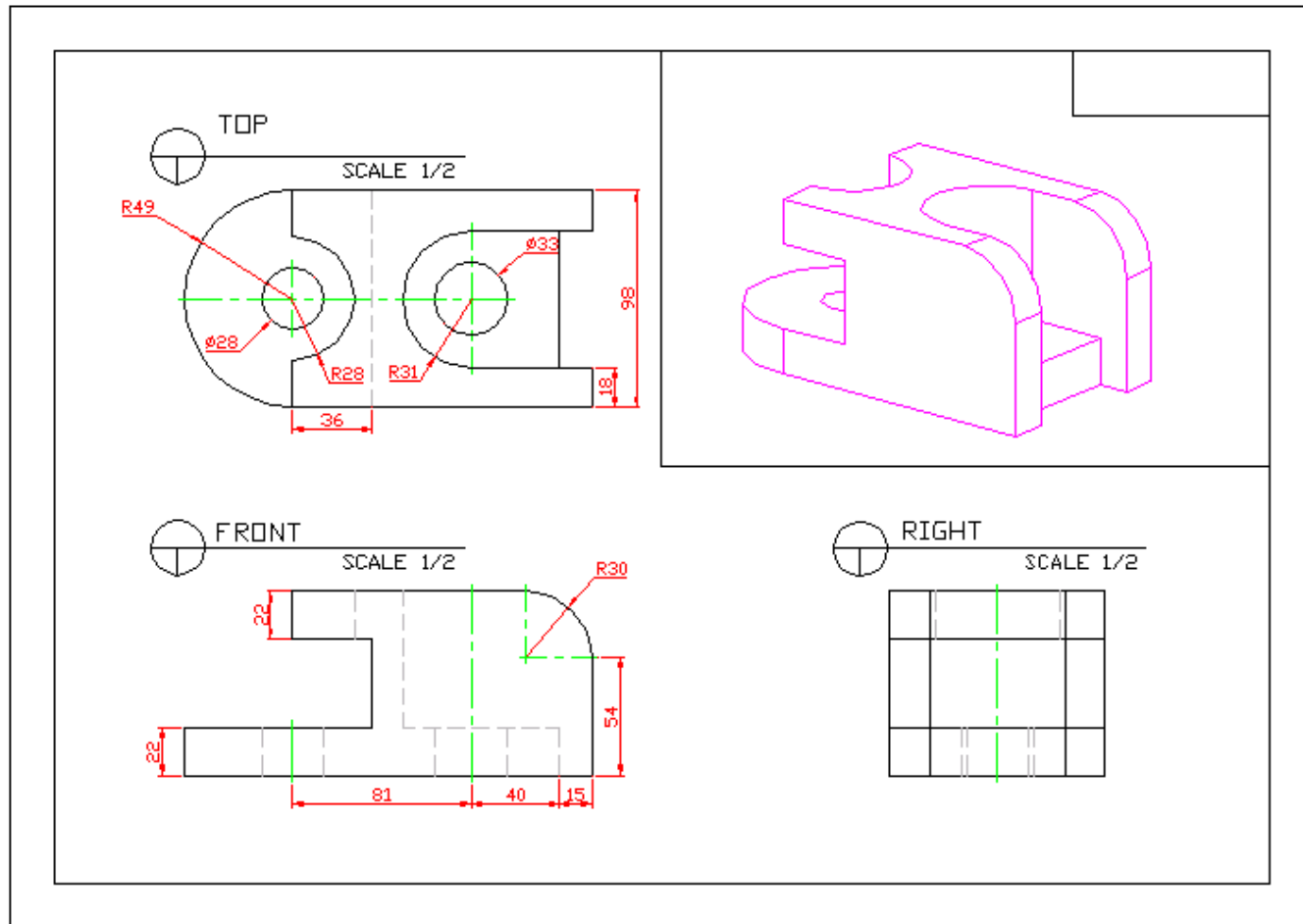
# V. PART DESIGN

## 13. 예 제도면 (4)



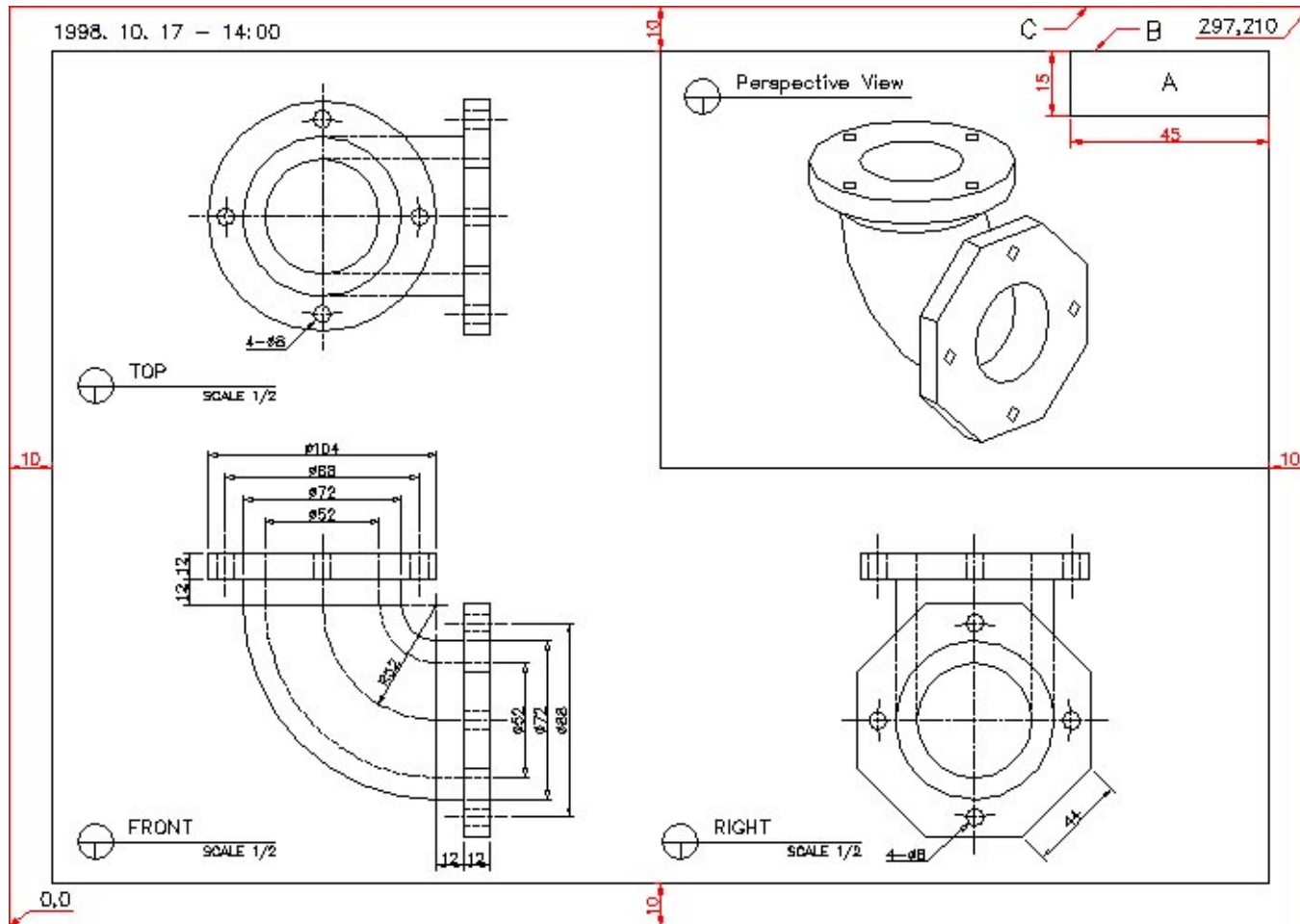
## V. PART DESIGN

### 13. 예 제도면 (5)



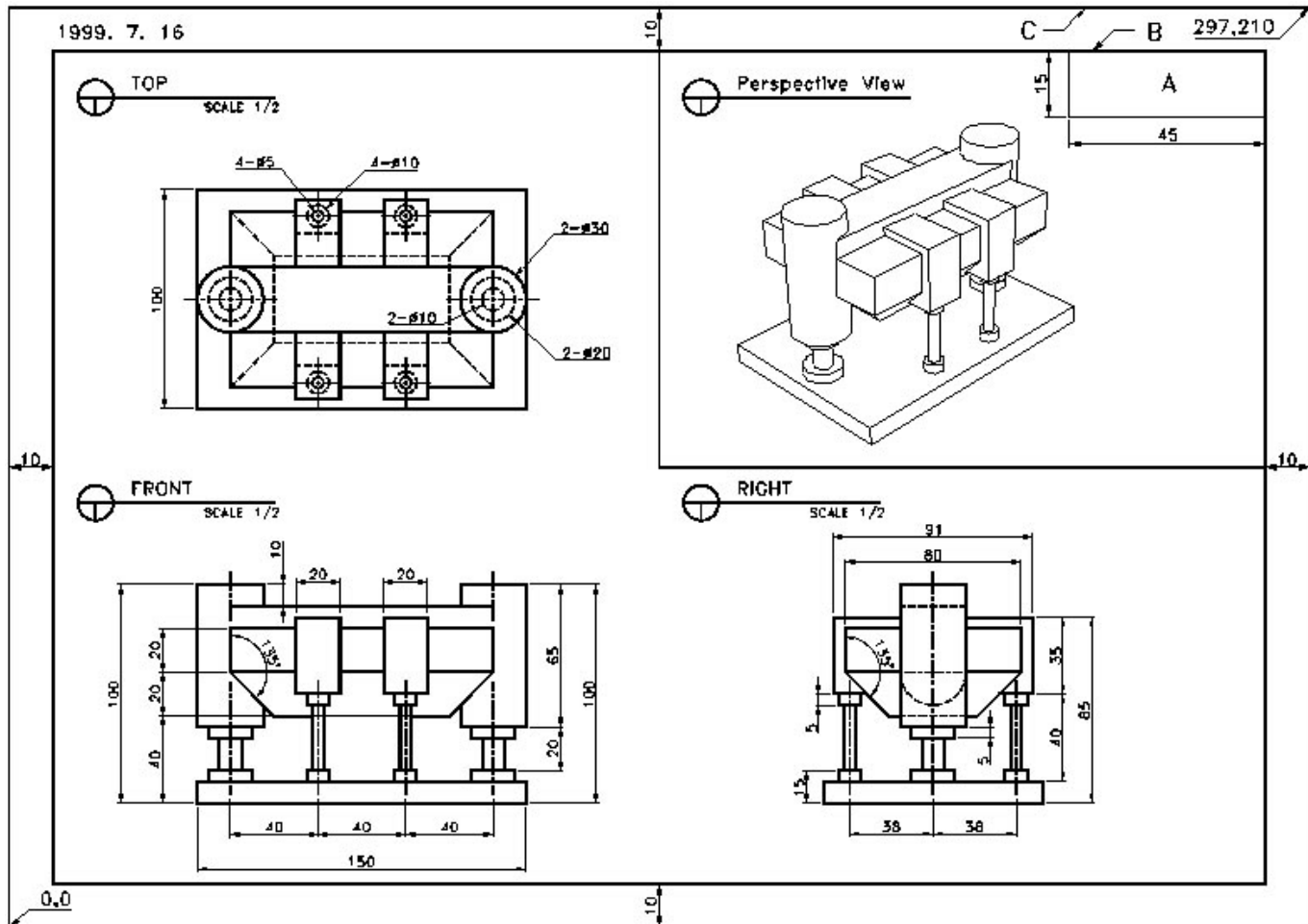
# V. PART DESIGN

## 13. 예 제도면 (6)



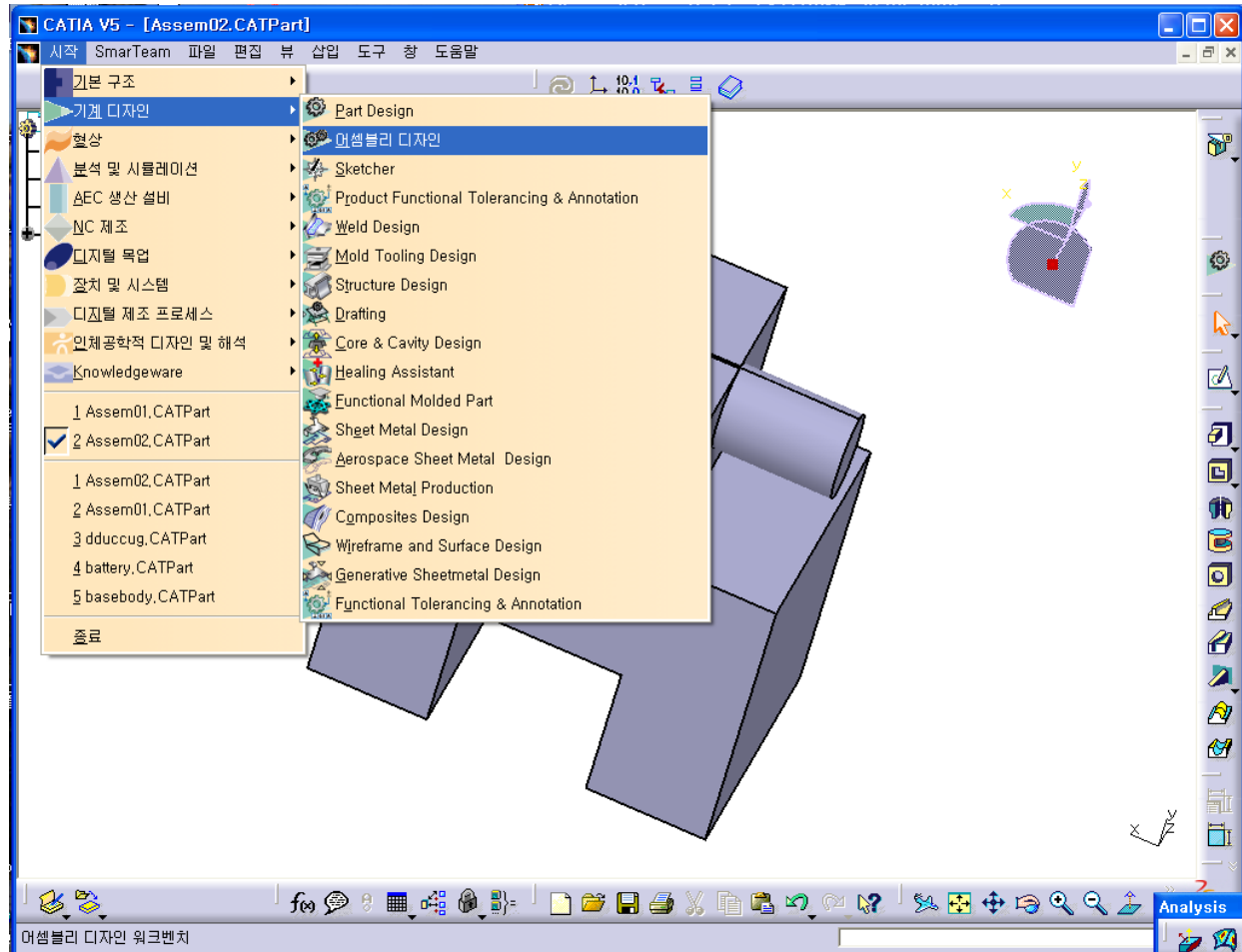
# V. PART DESIGN

## 13. 예 제도면 (7)



# VI. ASSEMBLY DESIGN

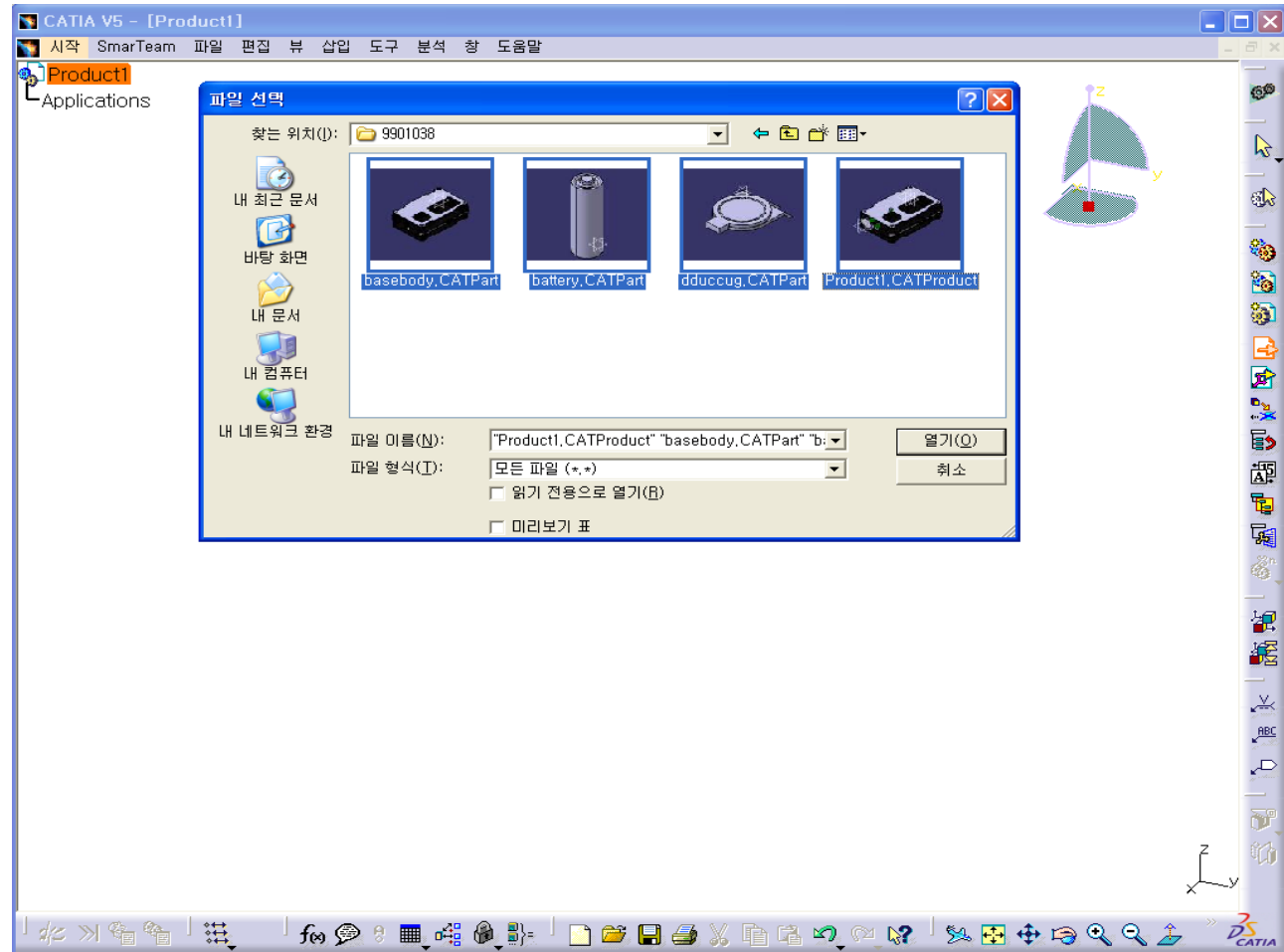
## 1. Introduction (1)





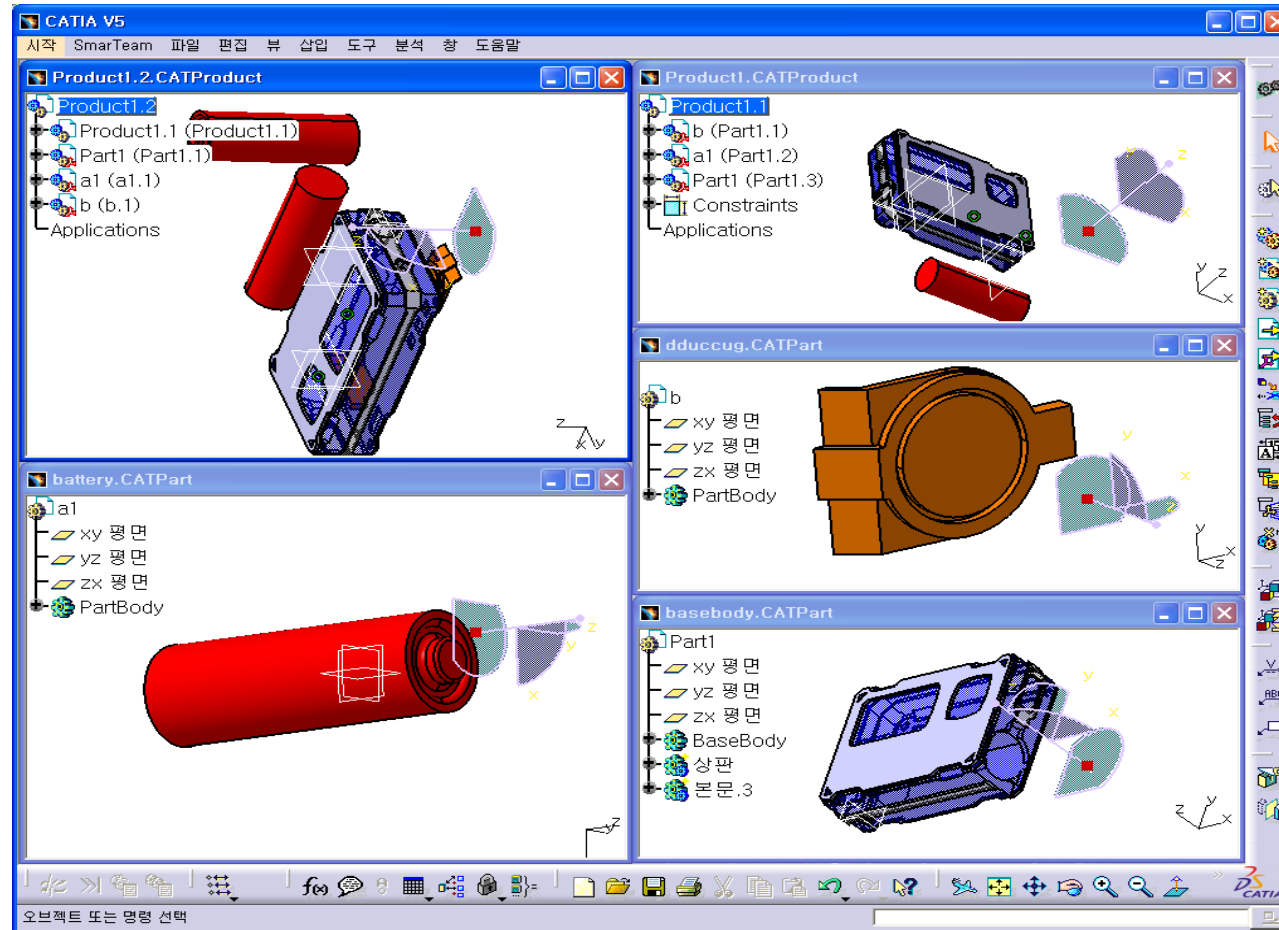
# VI. ASSEMBLY DESIGN

## 1. Introduction (2)



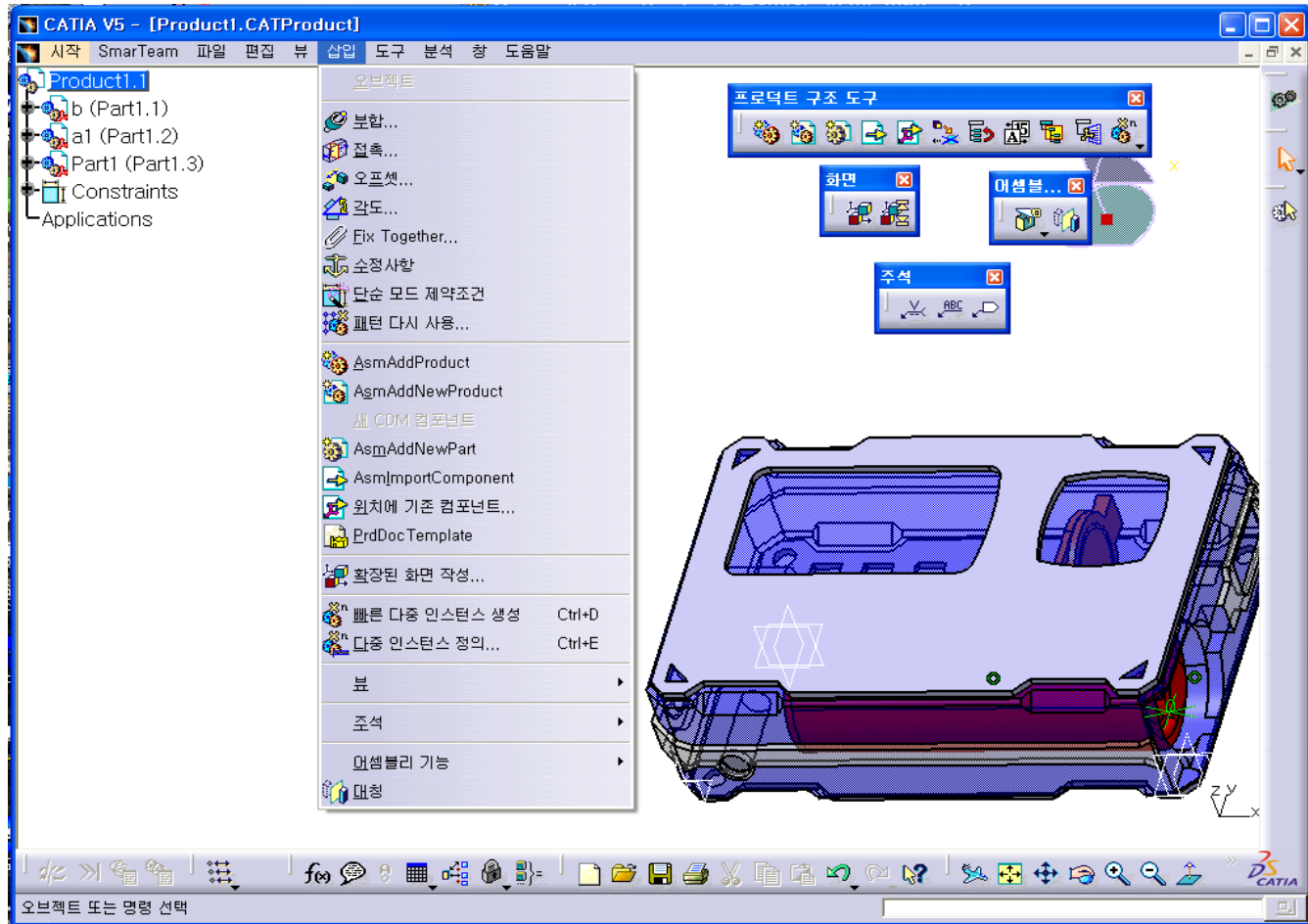
# VI. ASSEMBLY DESIGN

## 1. Introduction (3)



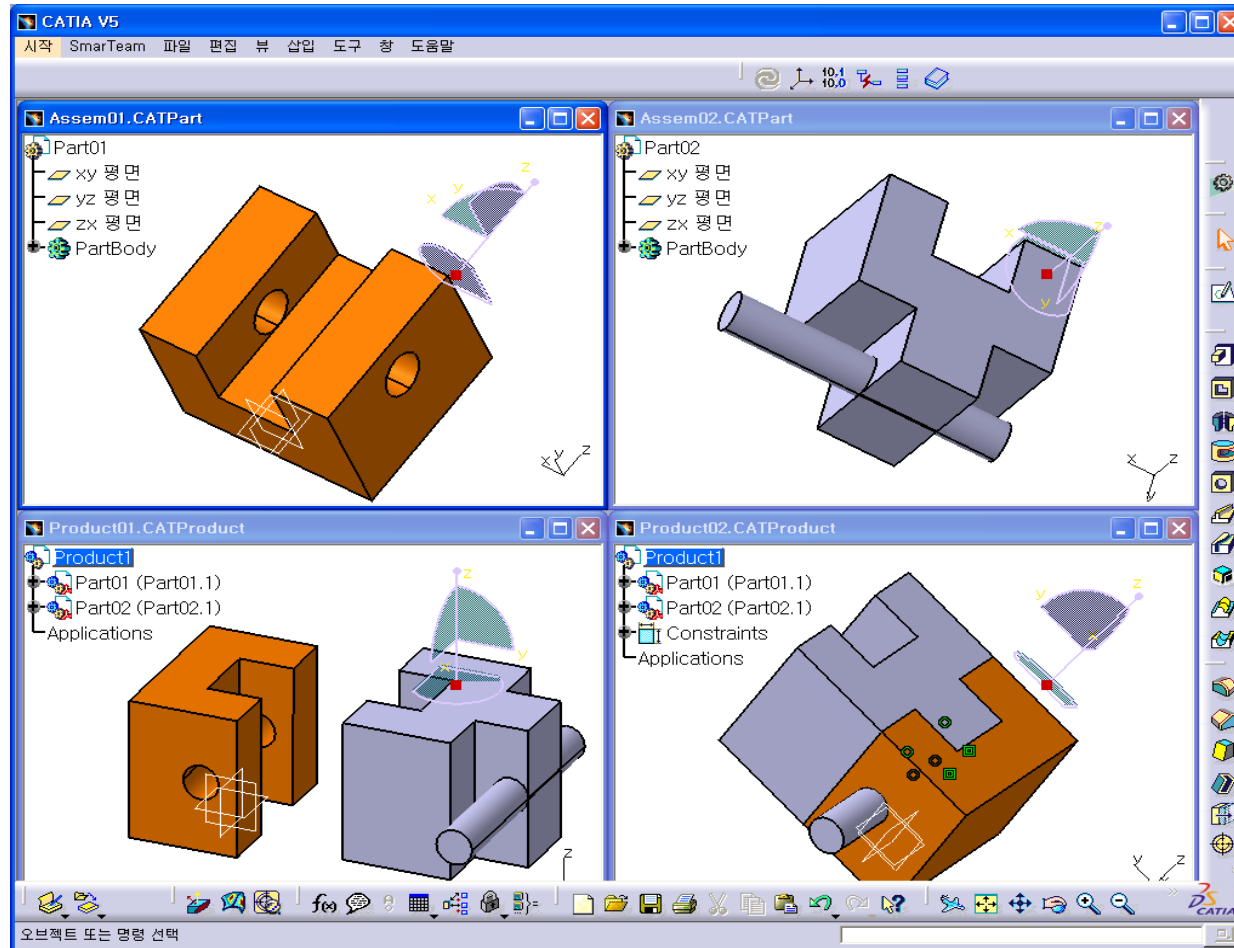
# VI. ASSEMBLY DESIGN

## 1. Introduction (4)



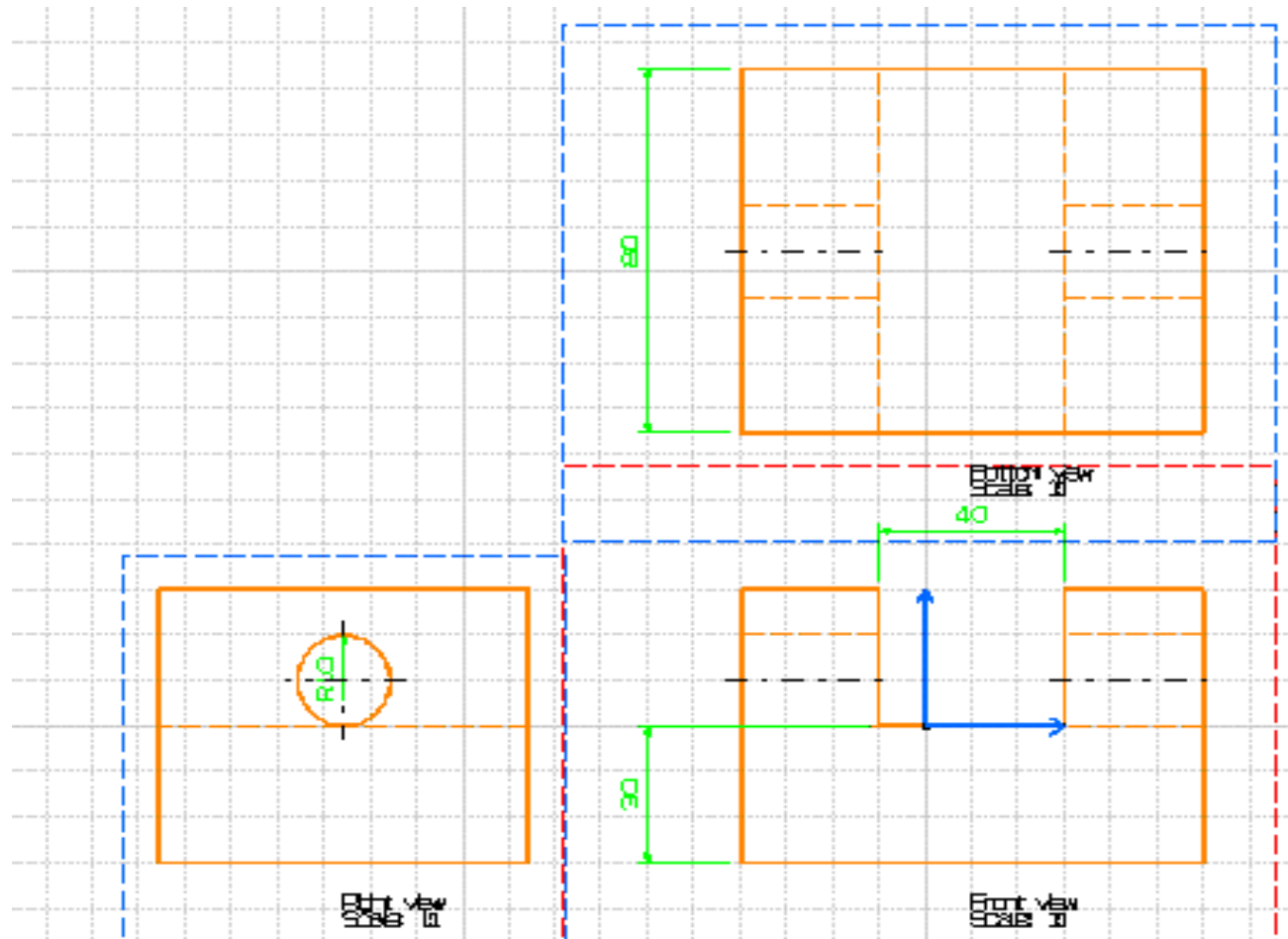
# VI. ASSEMBLY DESIGN

## 2. 실습 과제



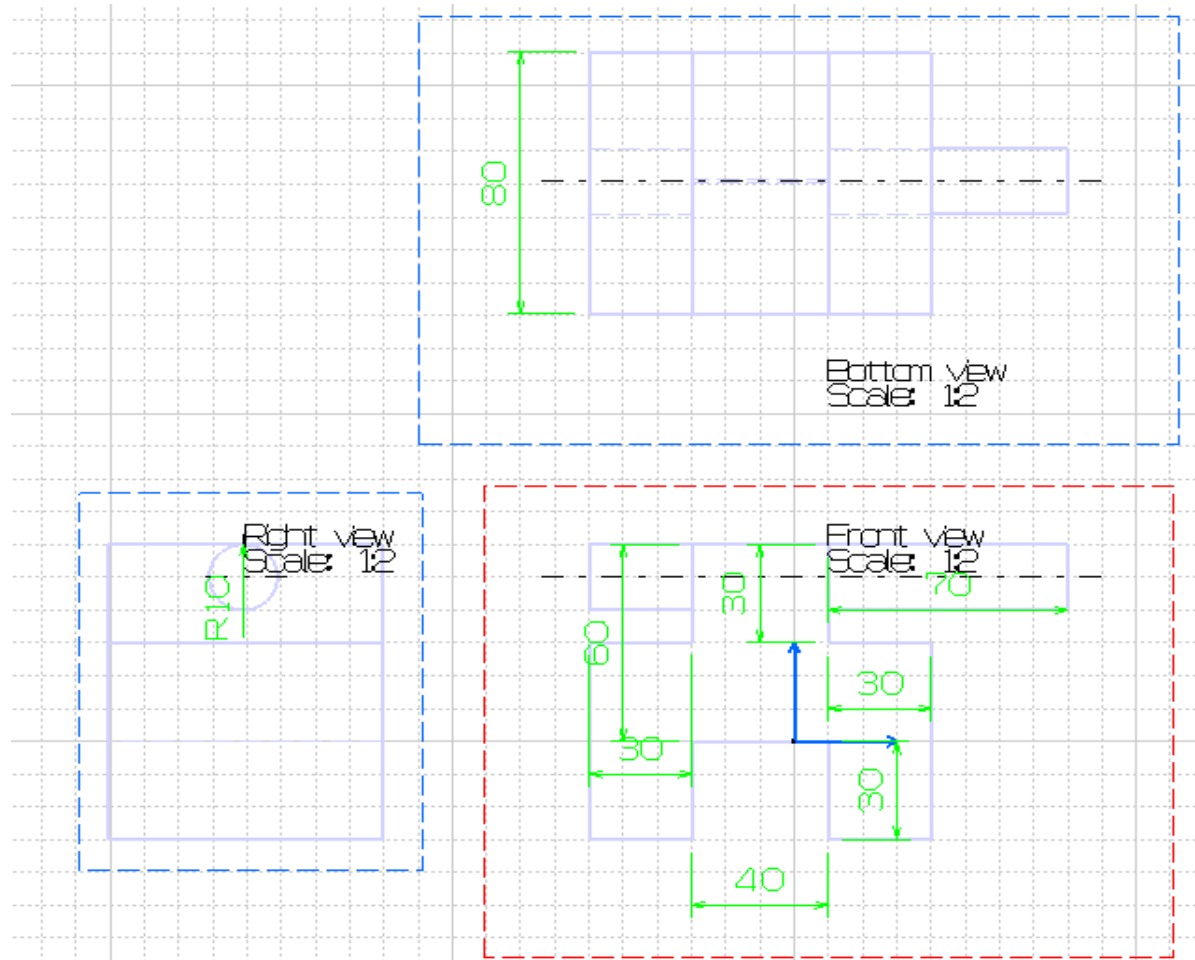
## VI. ASSEMBLY DESIGN

### 3. 모델링 (1)



## VI. ASSEMBLY DESIGN

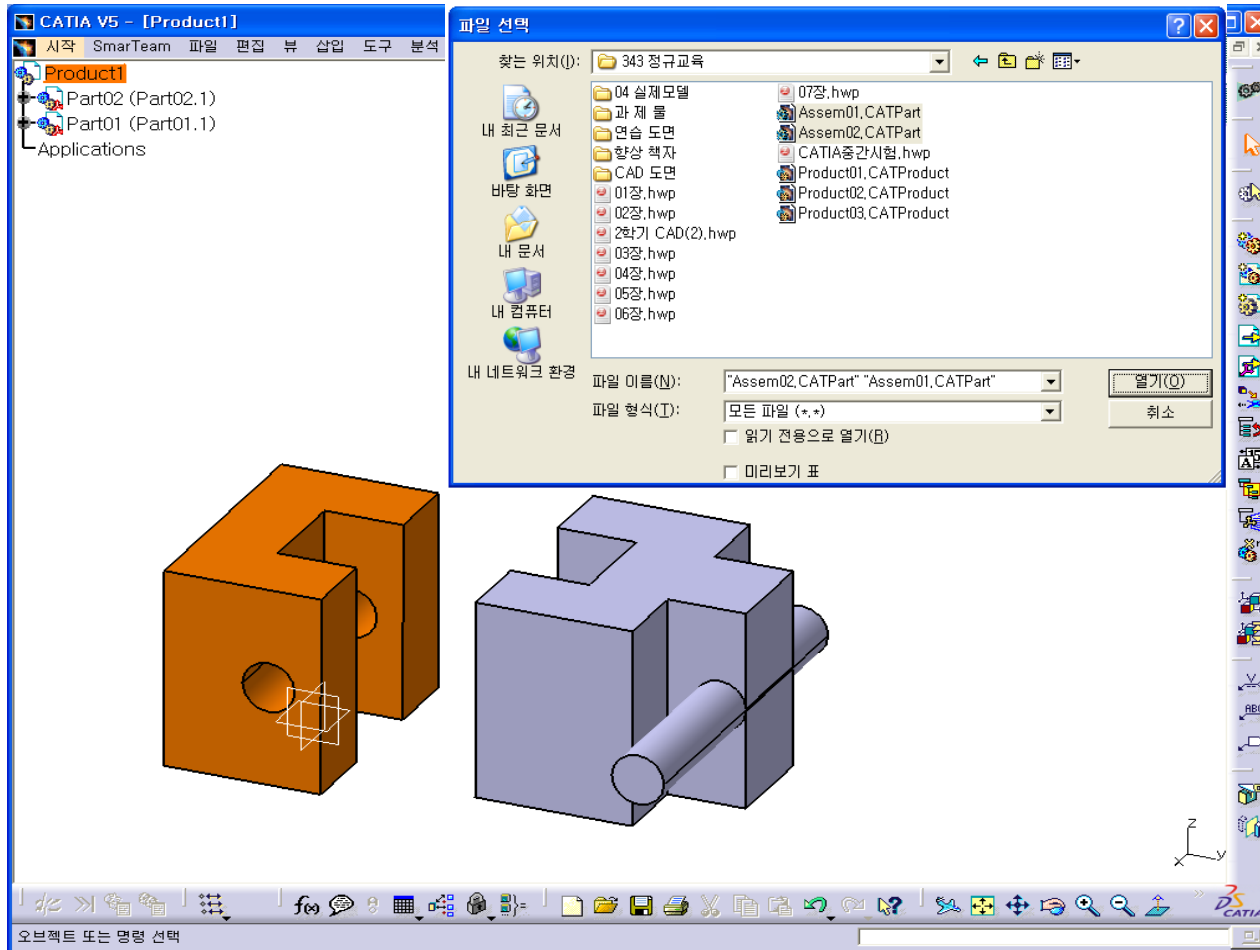
### 3. 모델링 (2)





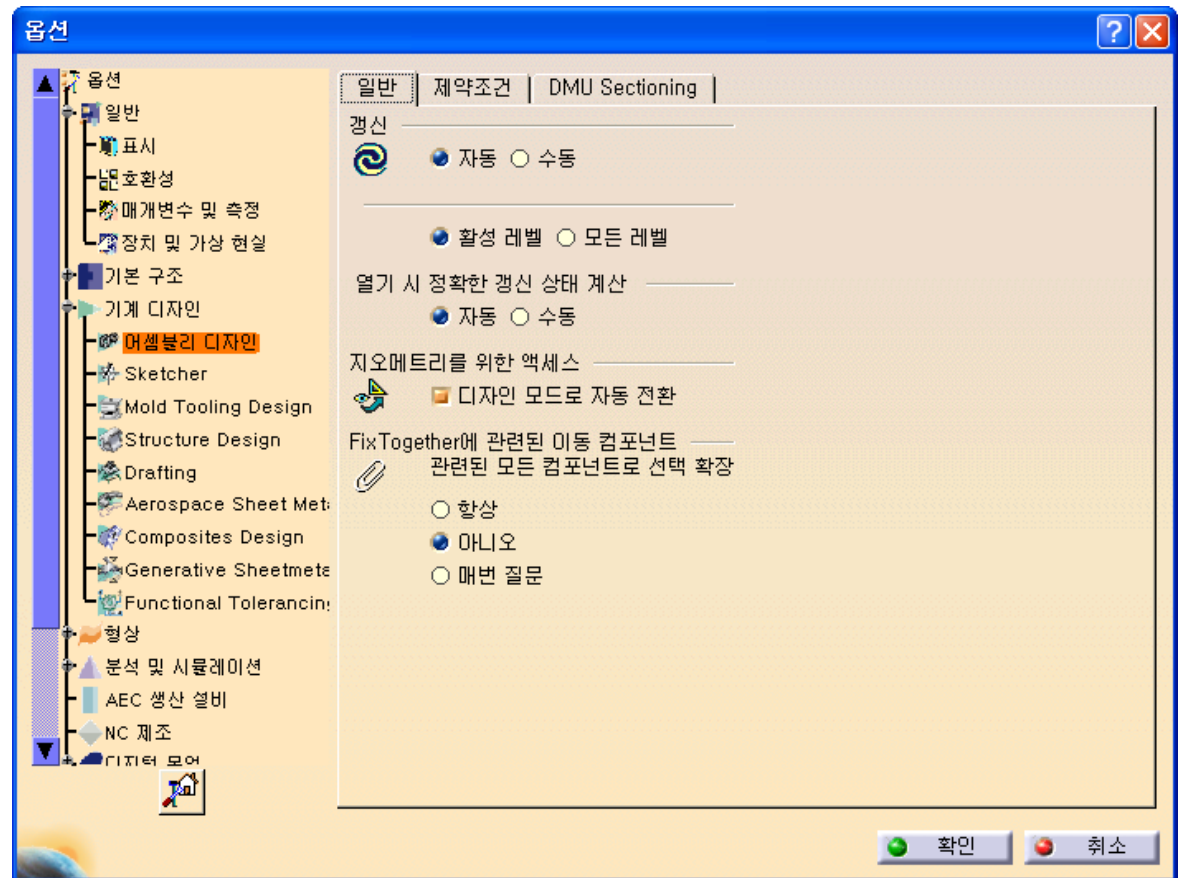
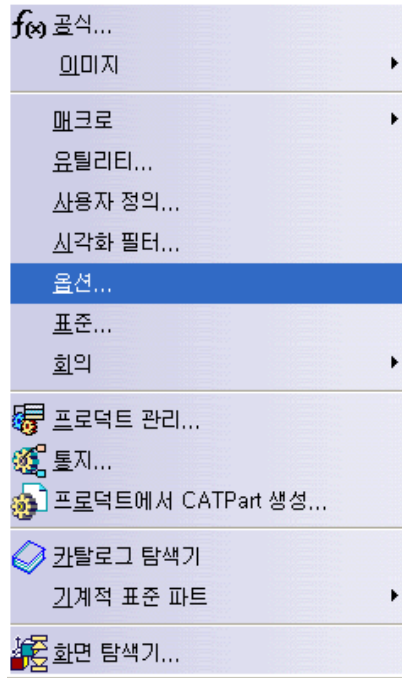
## VI. ASSEMBLY DESIGN

## 4. 어셈블리 (1)



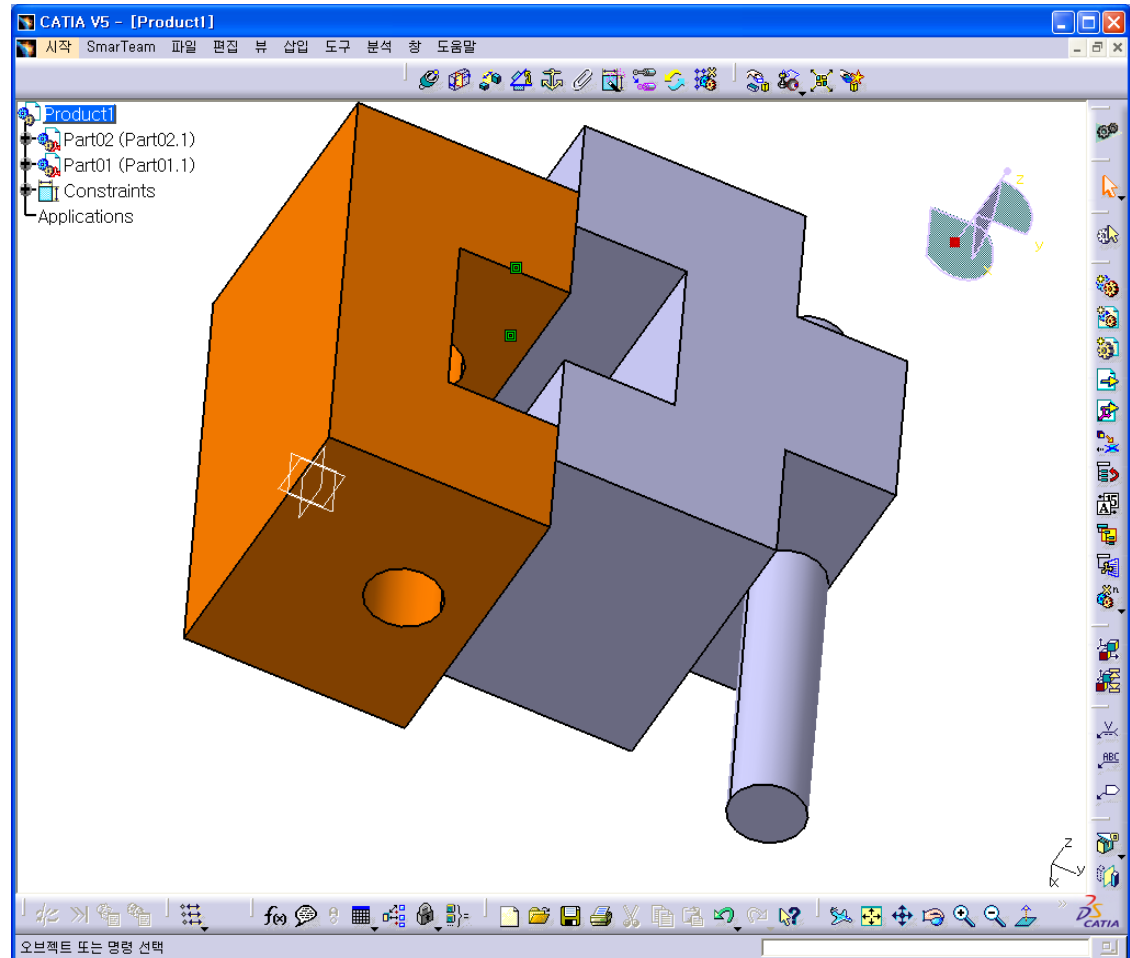
# VI. ASSEMBLY DESIGN

## 4. 어셈블리 (2)



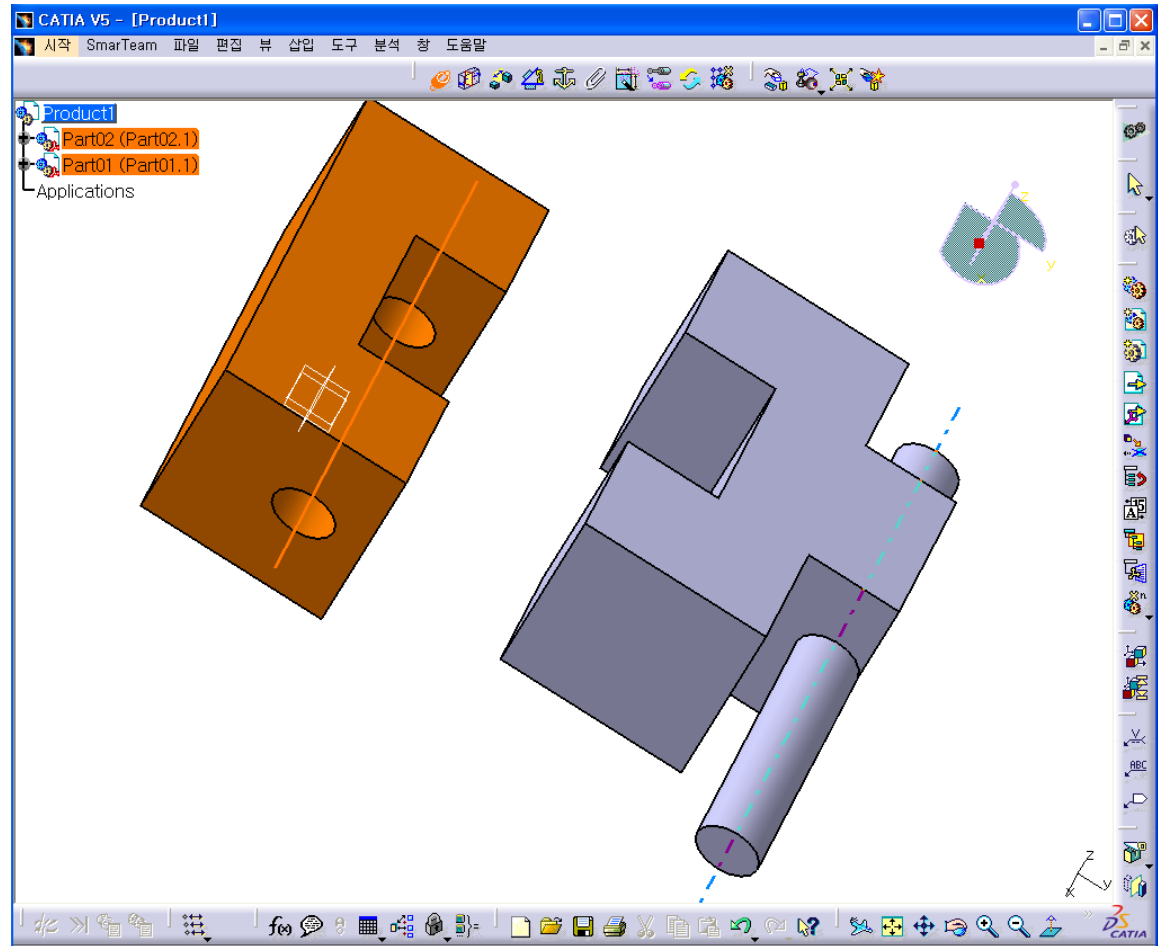
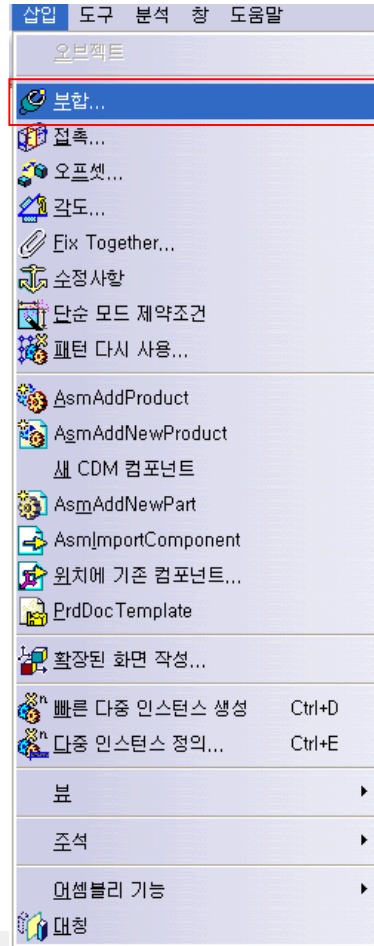
# VI. ASSEMBLY DESIGN

## 4. 어셈블리 (3)



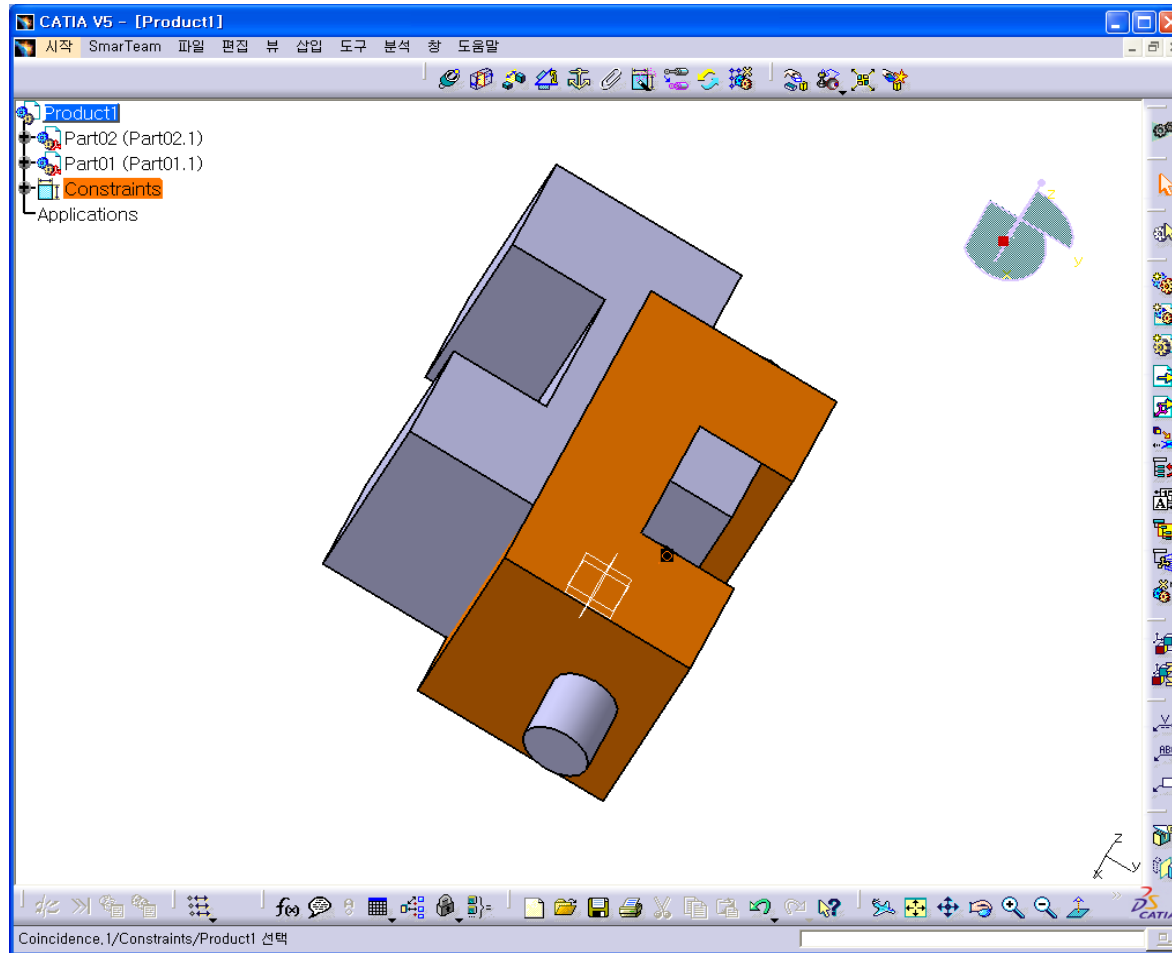
# VI. ASSEMBLY DESIGN

## 4. 어셈블리 (4)



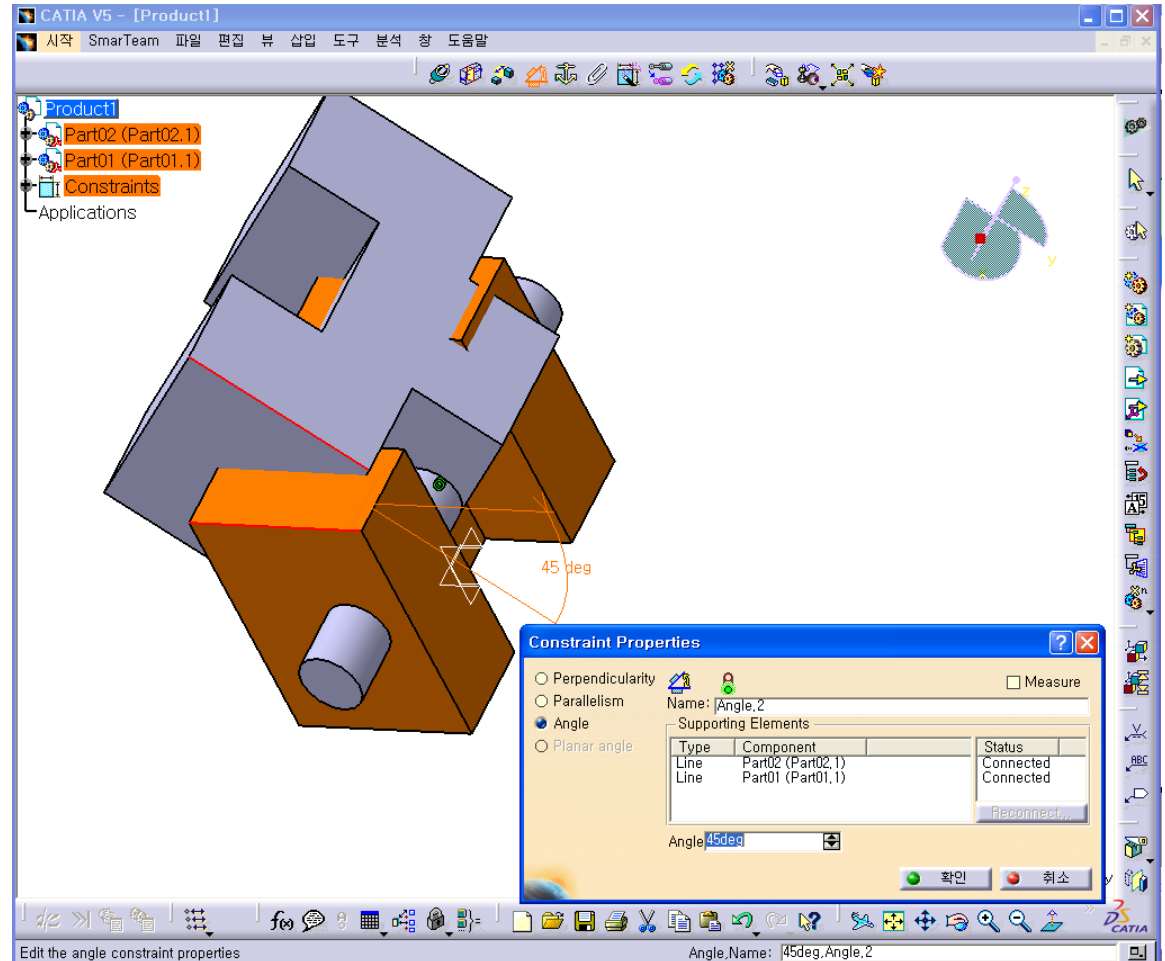
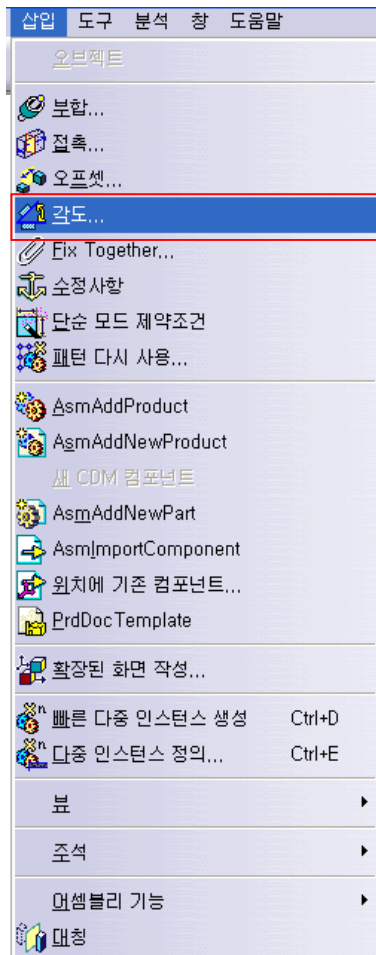
## VI. ASSEMBLY DESIGN

### 4. 어셈블리 (5)



# VI. ASSEMBLY DESIGN

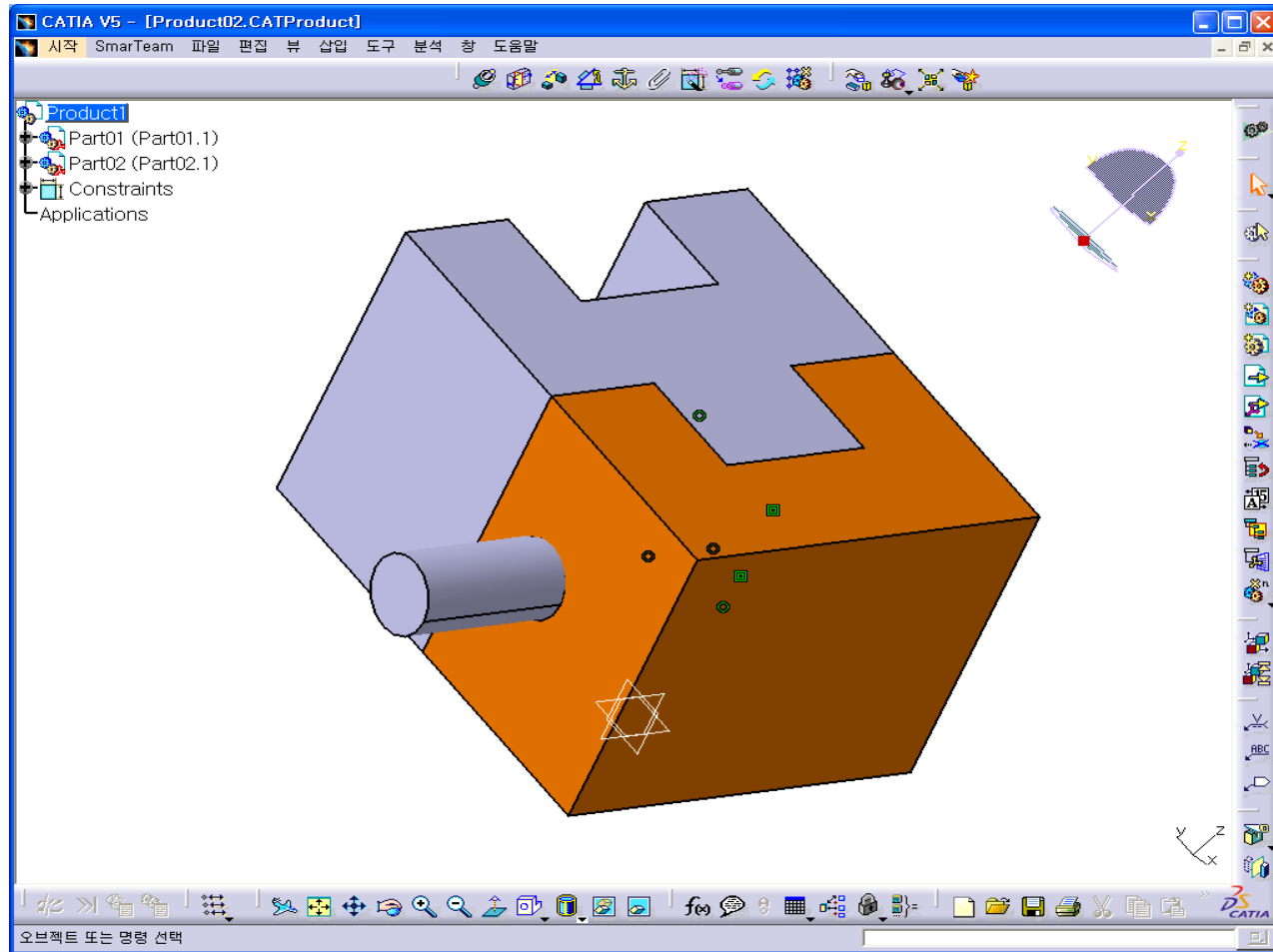
## 4. 어셈블리 (6)





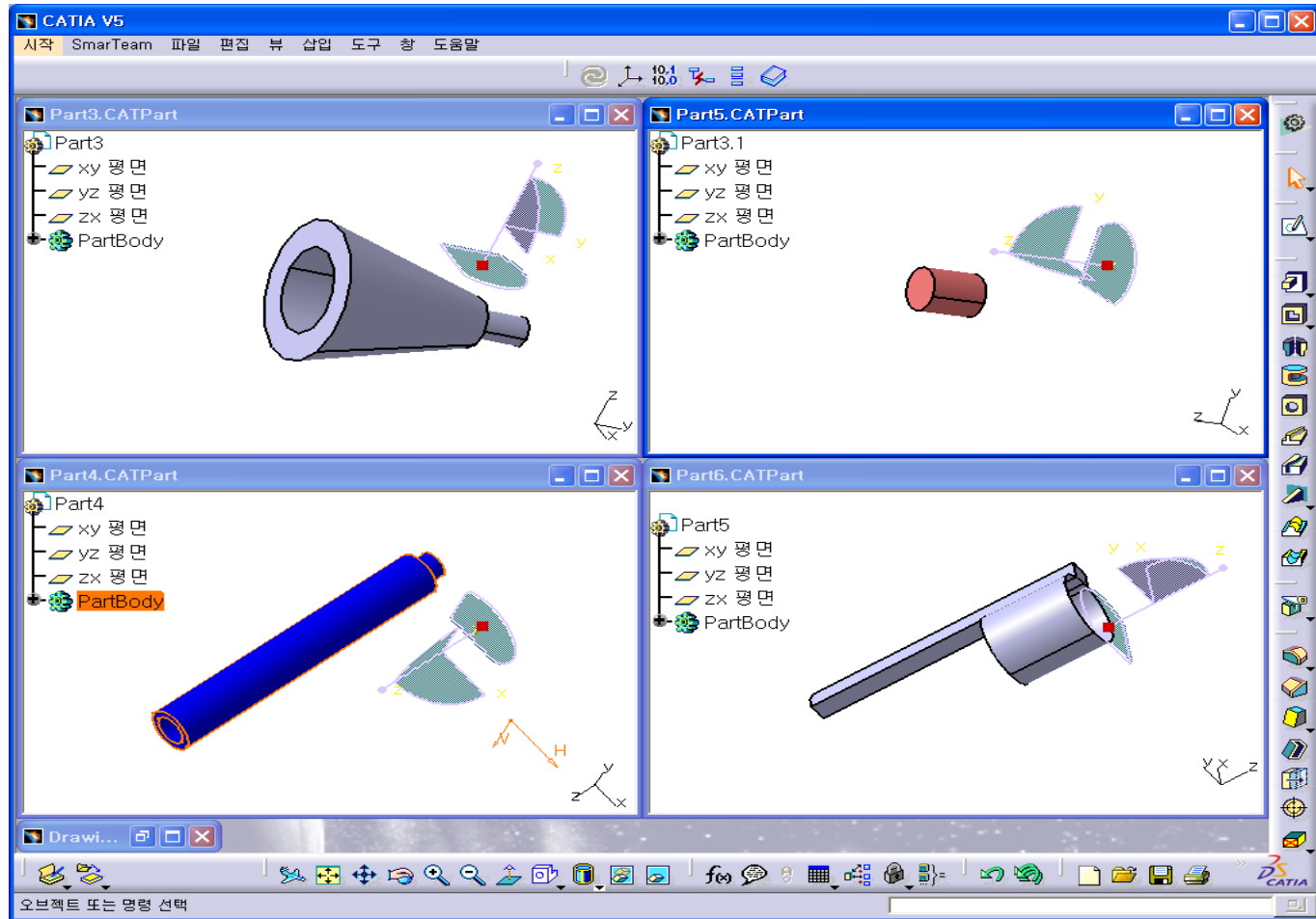
## VI. ASSEMBLY DESIGN

### 4. 어셈블리 (7)



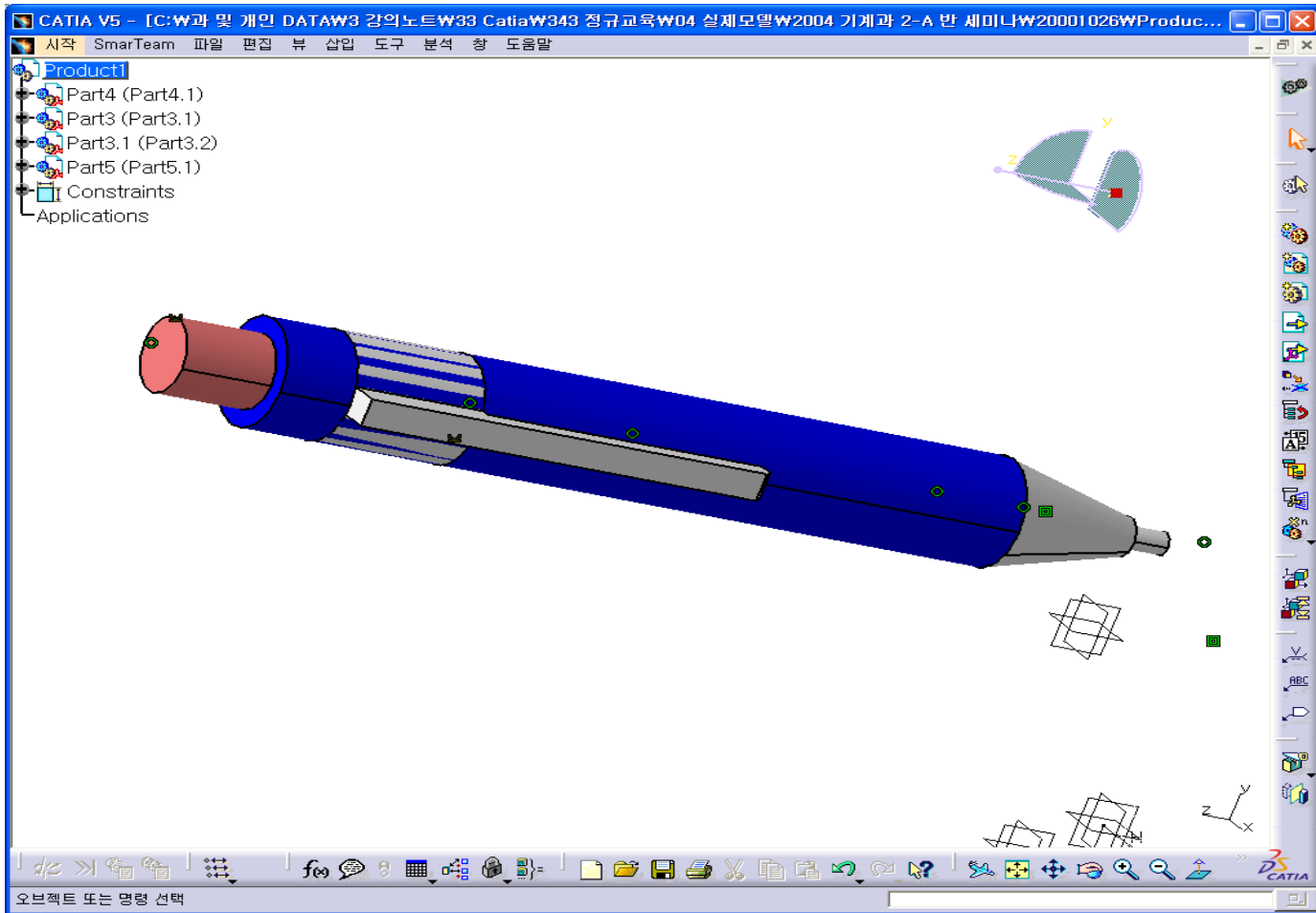
# VI. ASSEMBLY DESIGN

## 5. 어셈블리예제 (1)



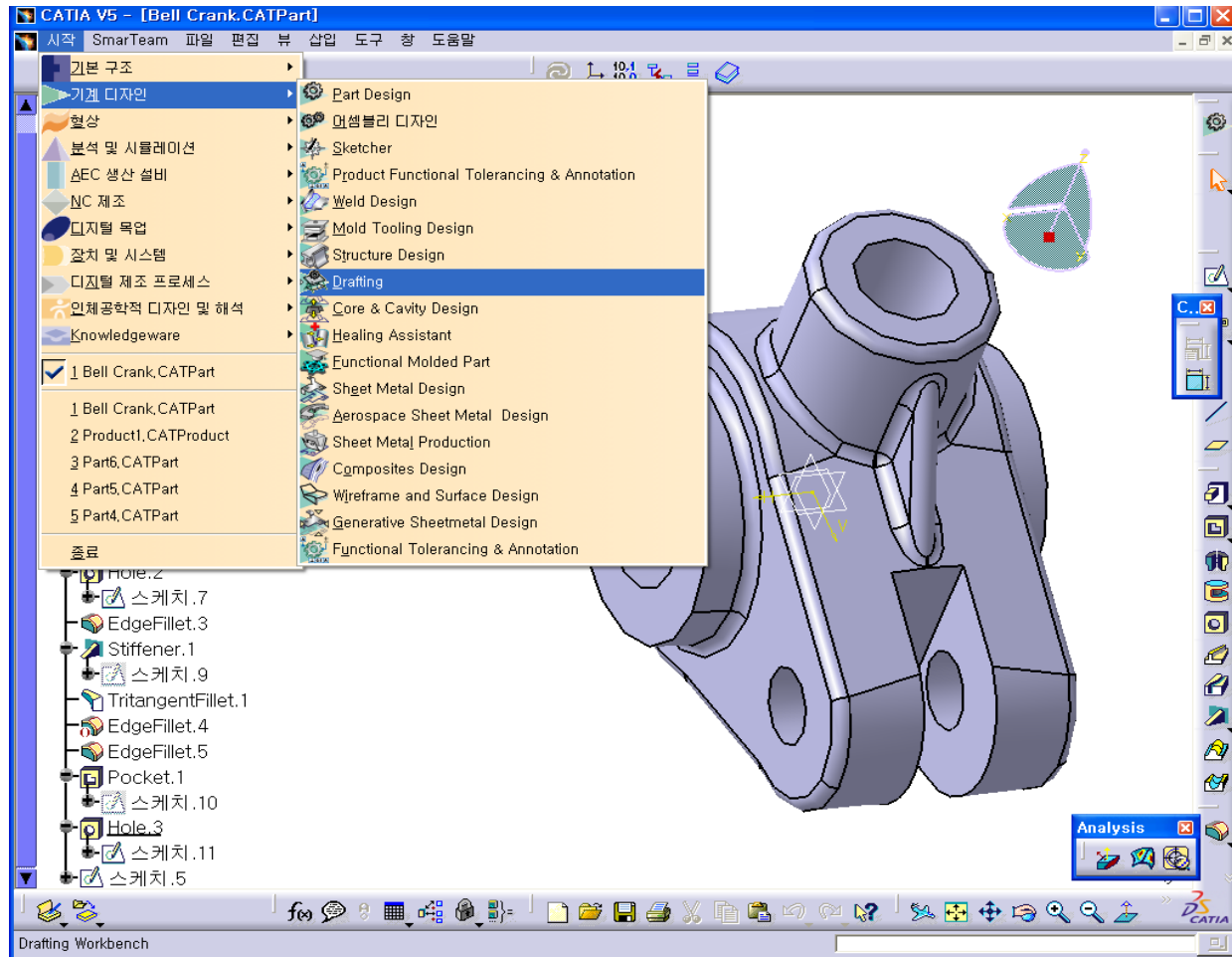
## VI. ASSEMBLY DESIGN

### 5. 어셈블리예제 (2)



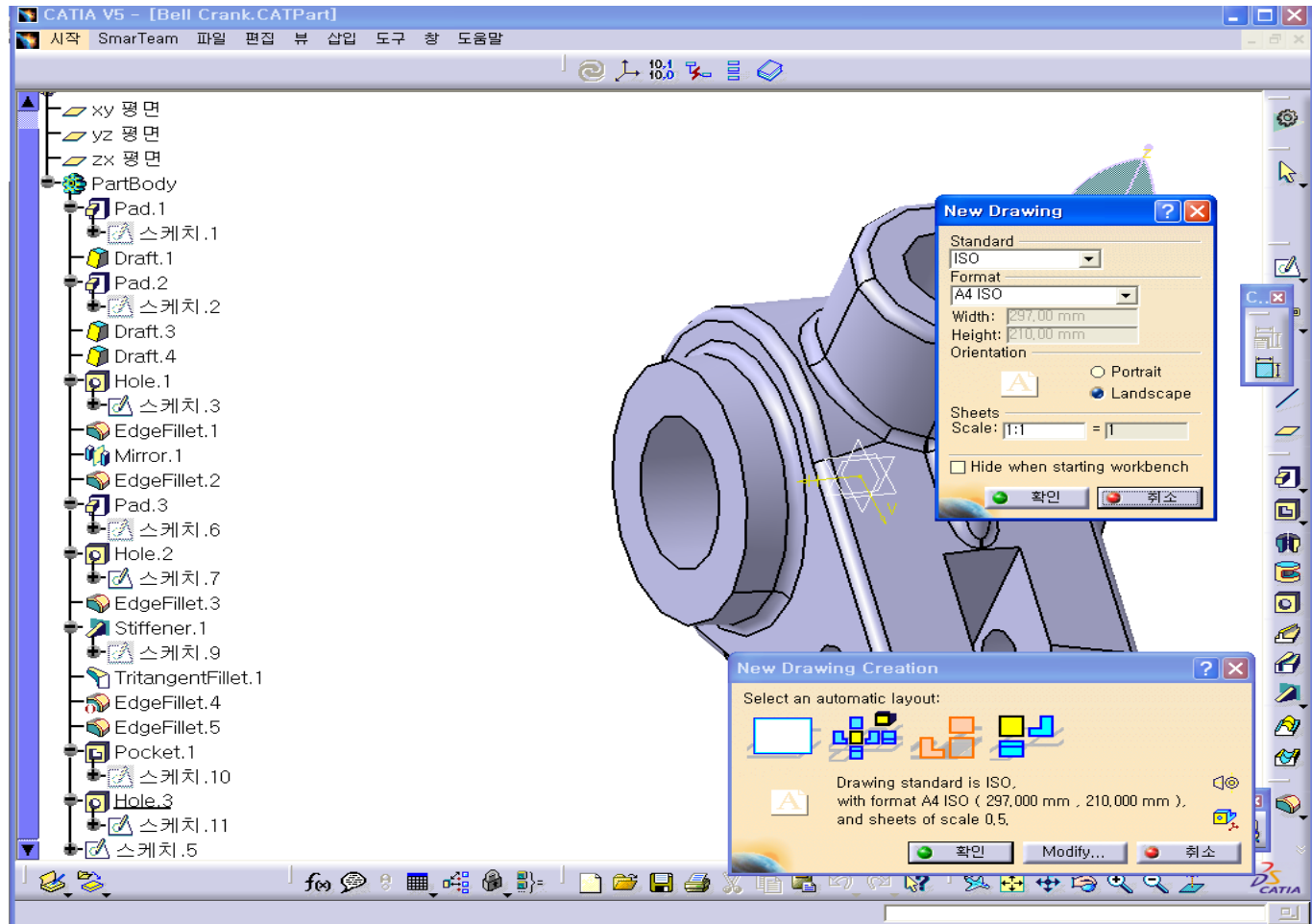
# VII. DRAFTING

## 1. Introduction (1)



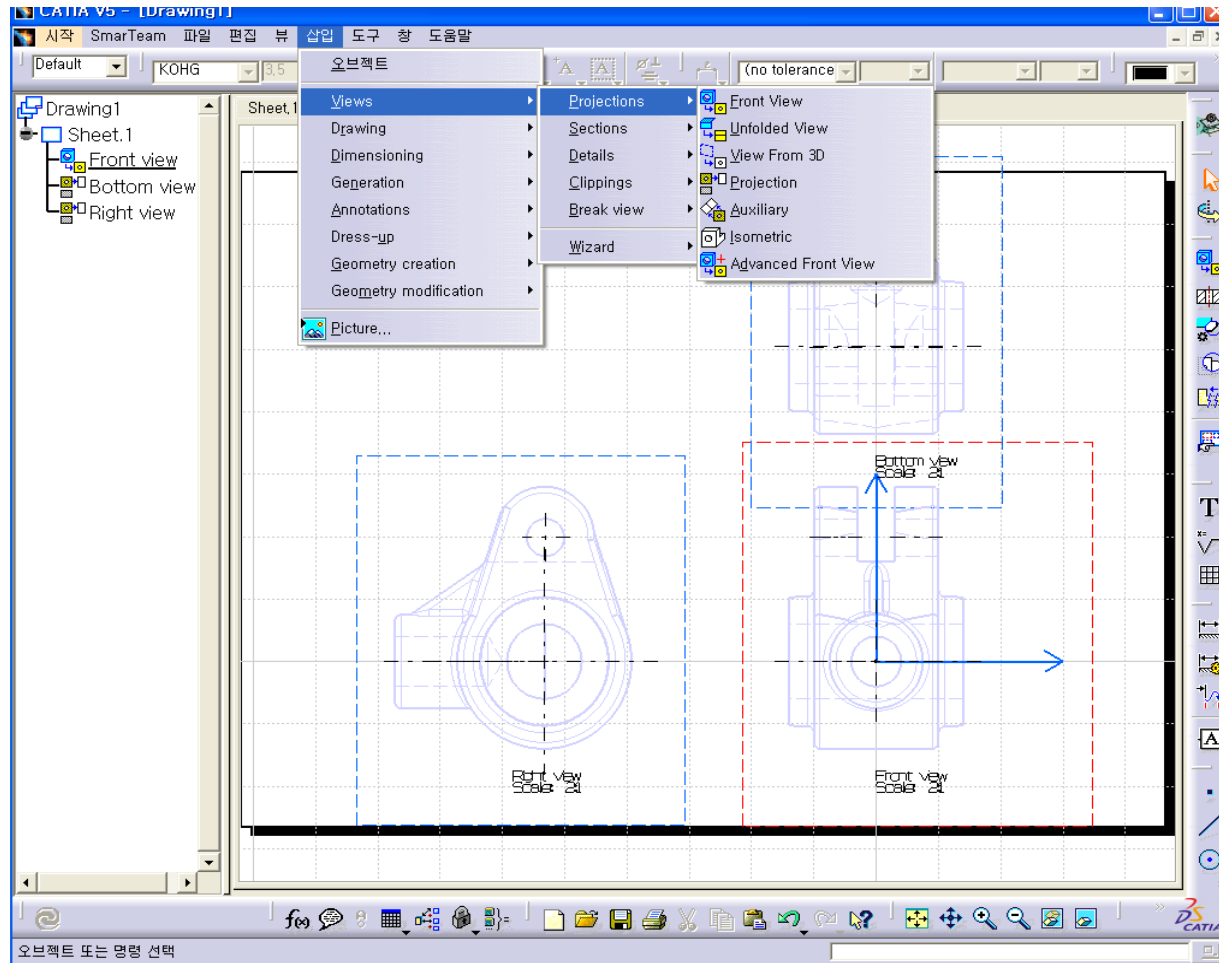
# VII. DRAFTING

## 1. Introduction (2)



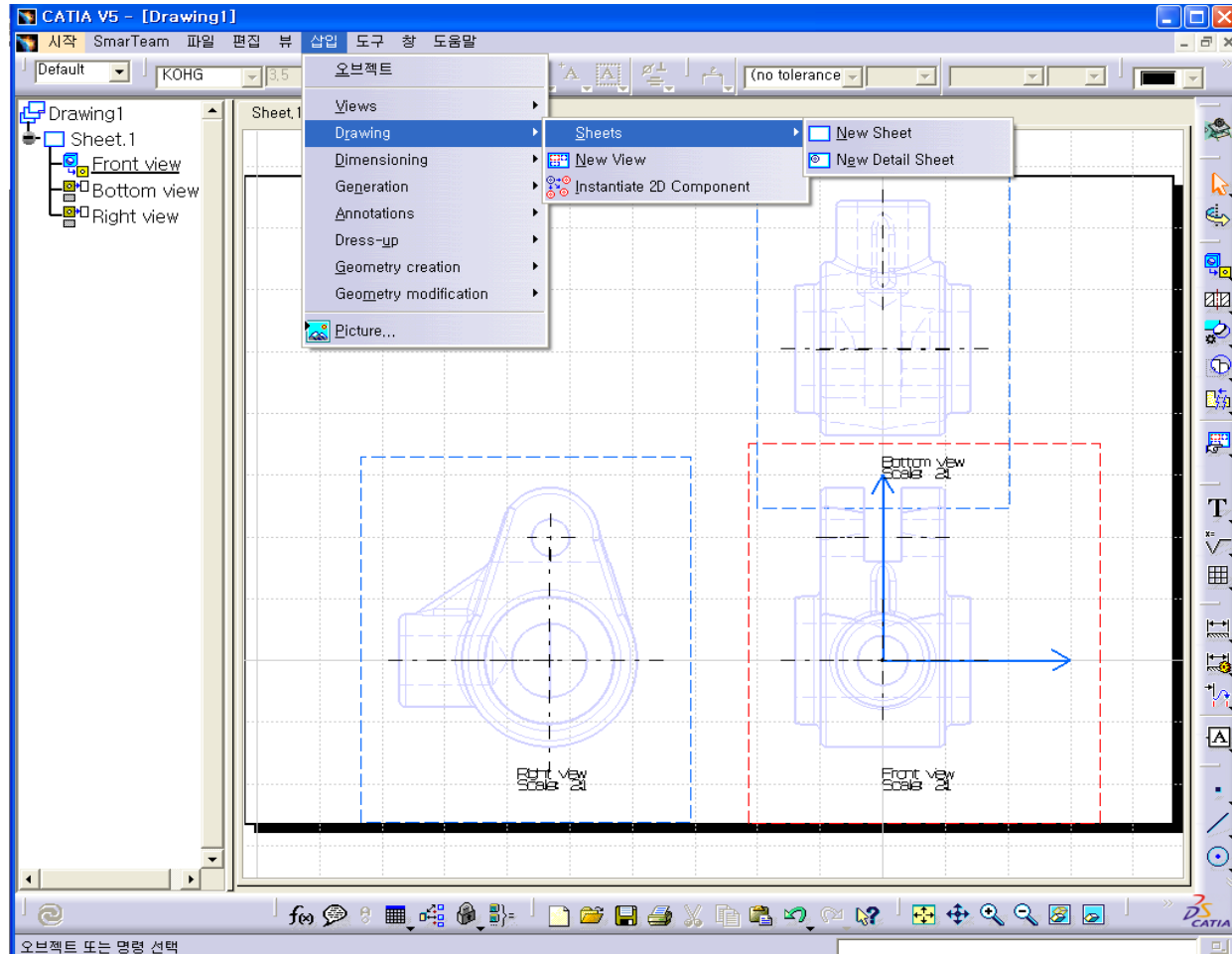
## VII. DRAFTING

### 1. Introduction (3)



# VII. DRAFTING

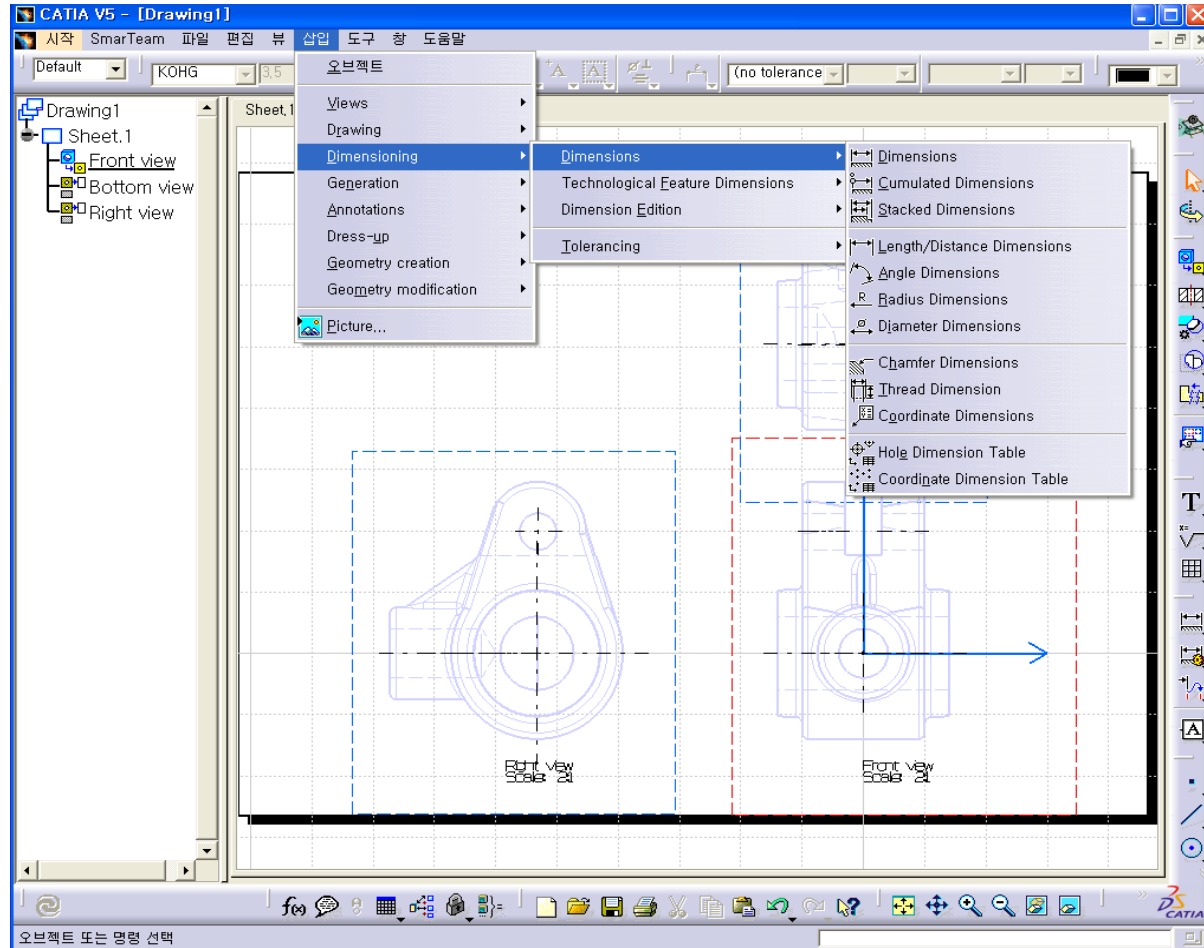
## 1. Introduction (4)





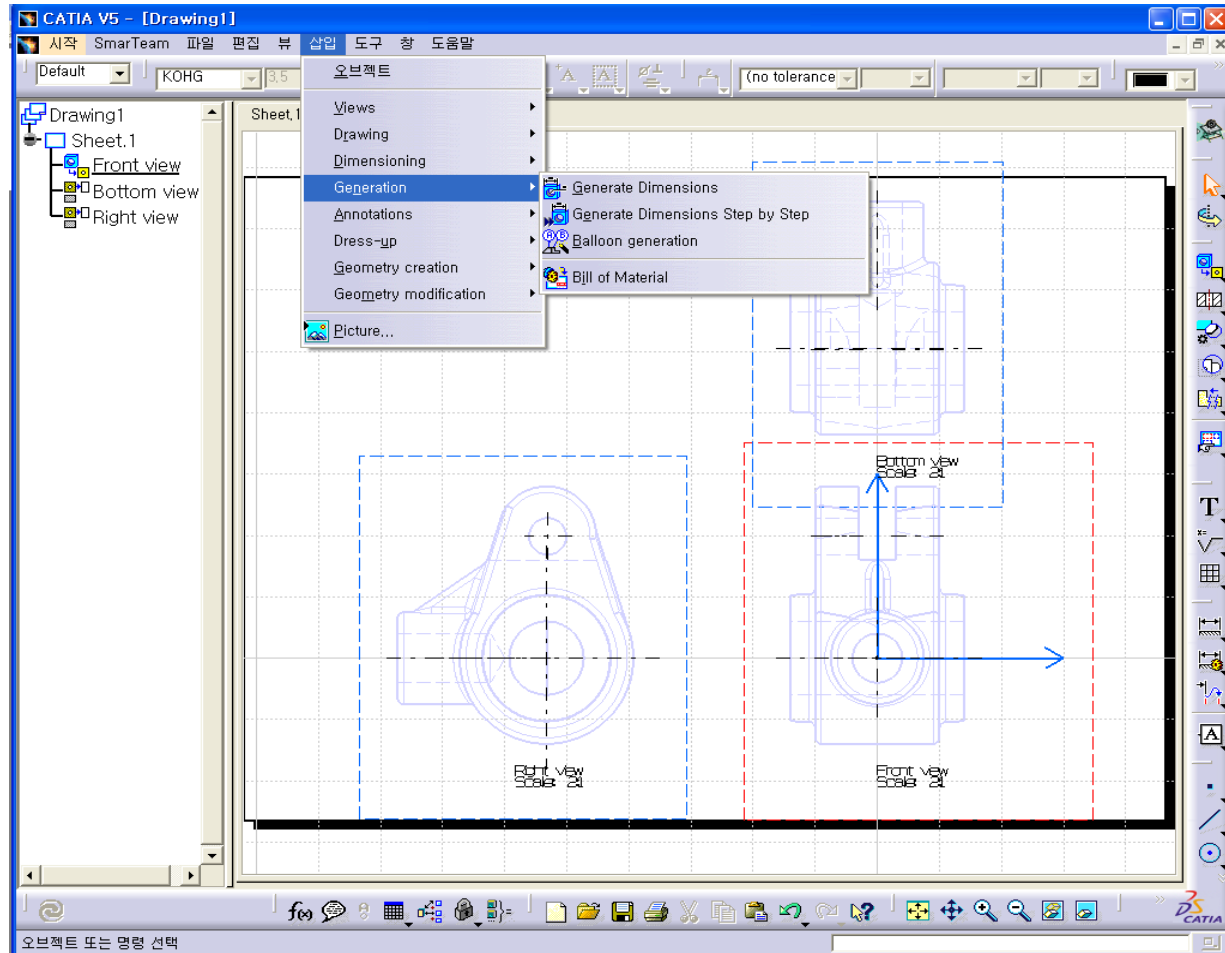
# VII. DRAFTING

## 1. Introduction (5)



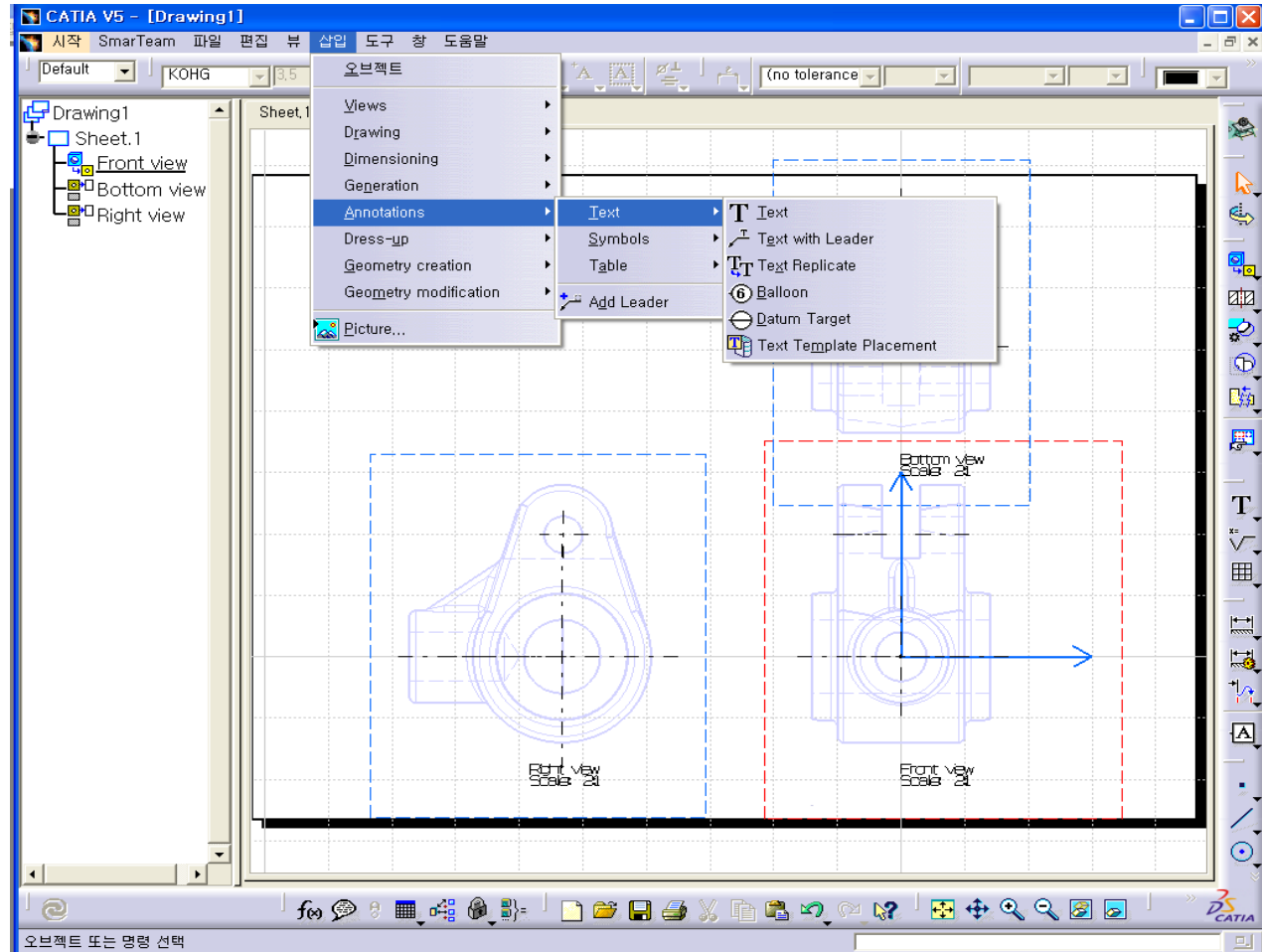
# VII. DRAFTING

## 1. Introduction (6)



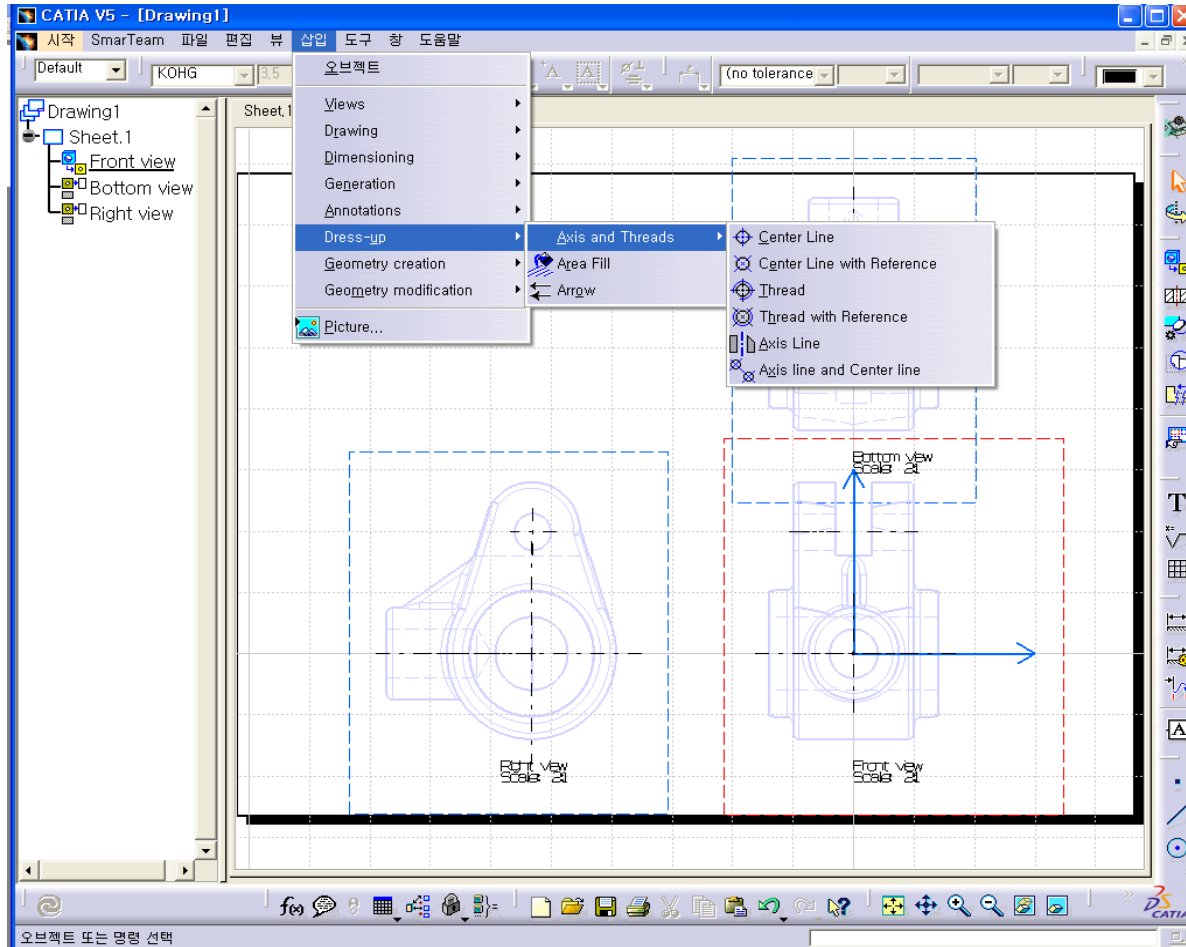
# VII. DRAFTING

## 1. Introduction (7)



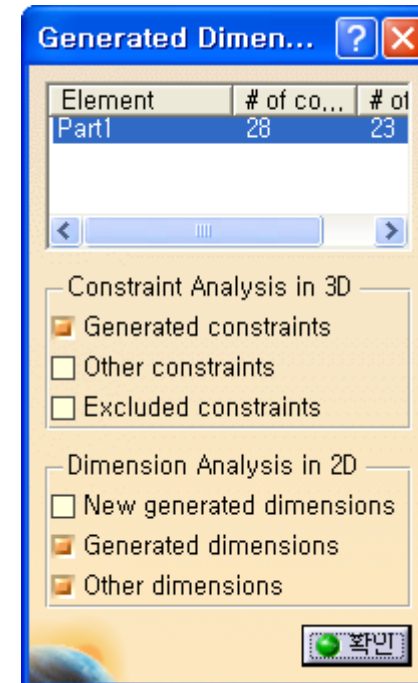
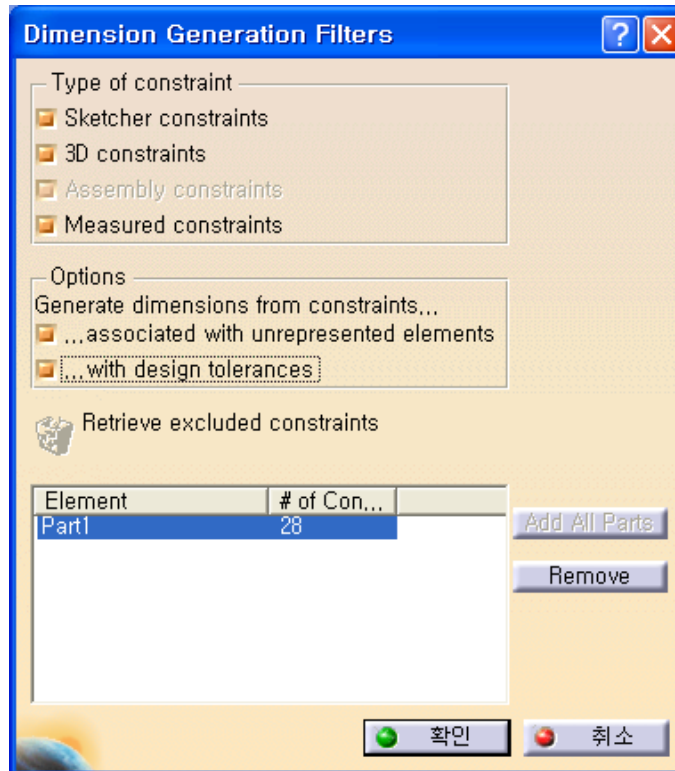
# VII. DRAFTING

## 1. Introduction (8)



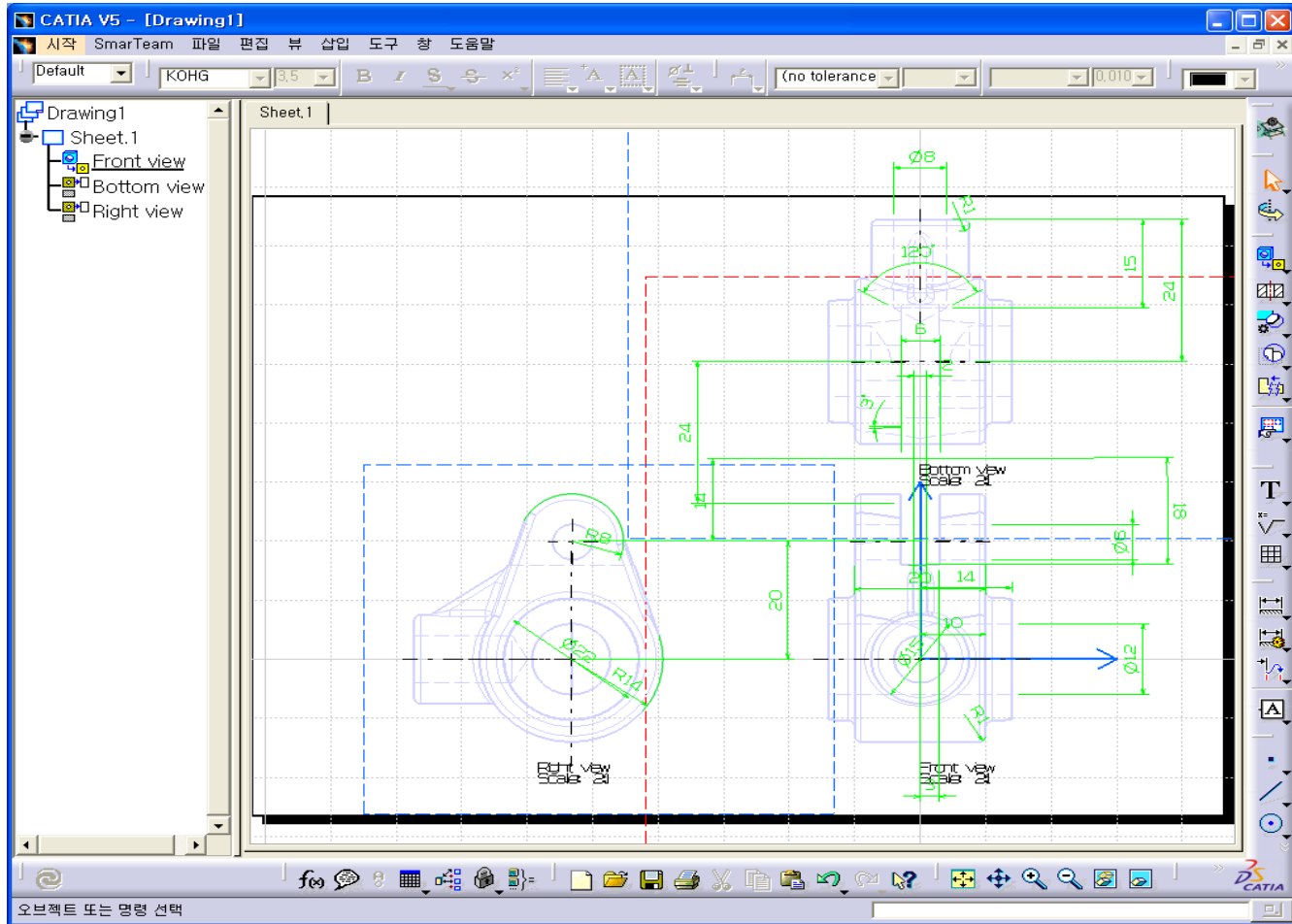
# VII. DRAFTING

## 2. 단일 부품 드래프팅 (1)



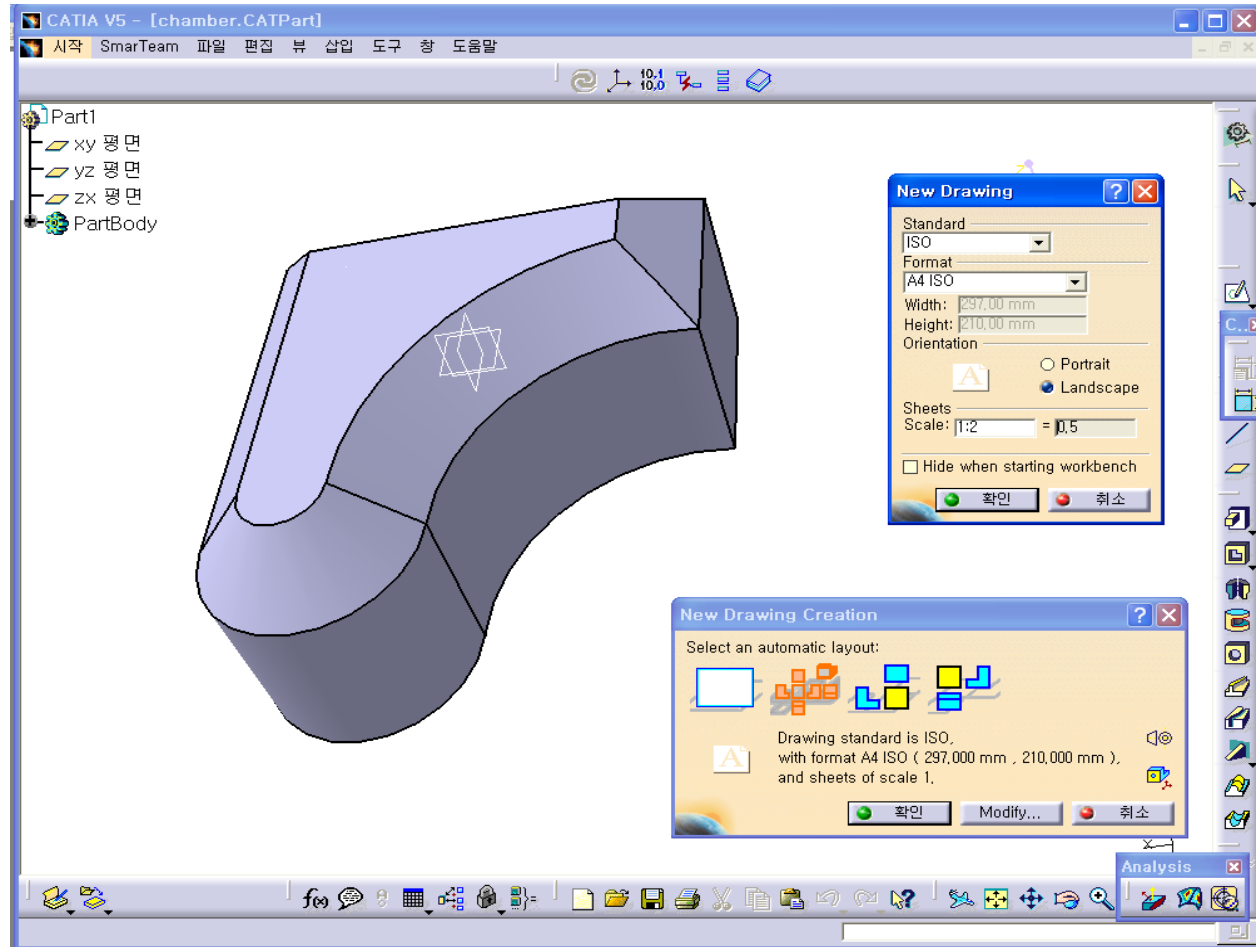
# VII. DRAFTING

## 2. 단일 부품 드래프팅 (2)



# VII. DRAFTING

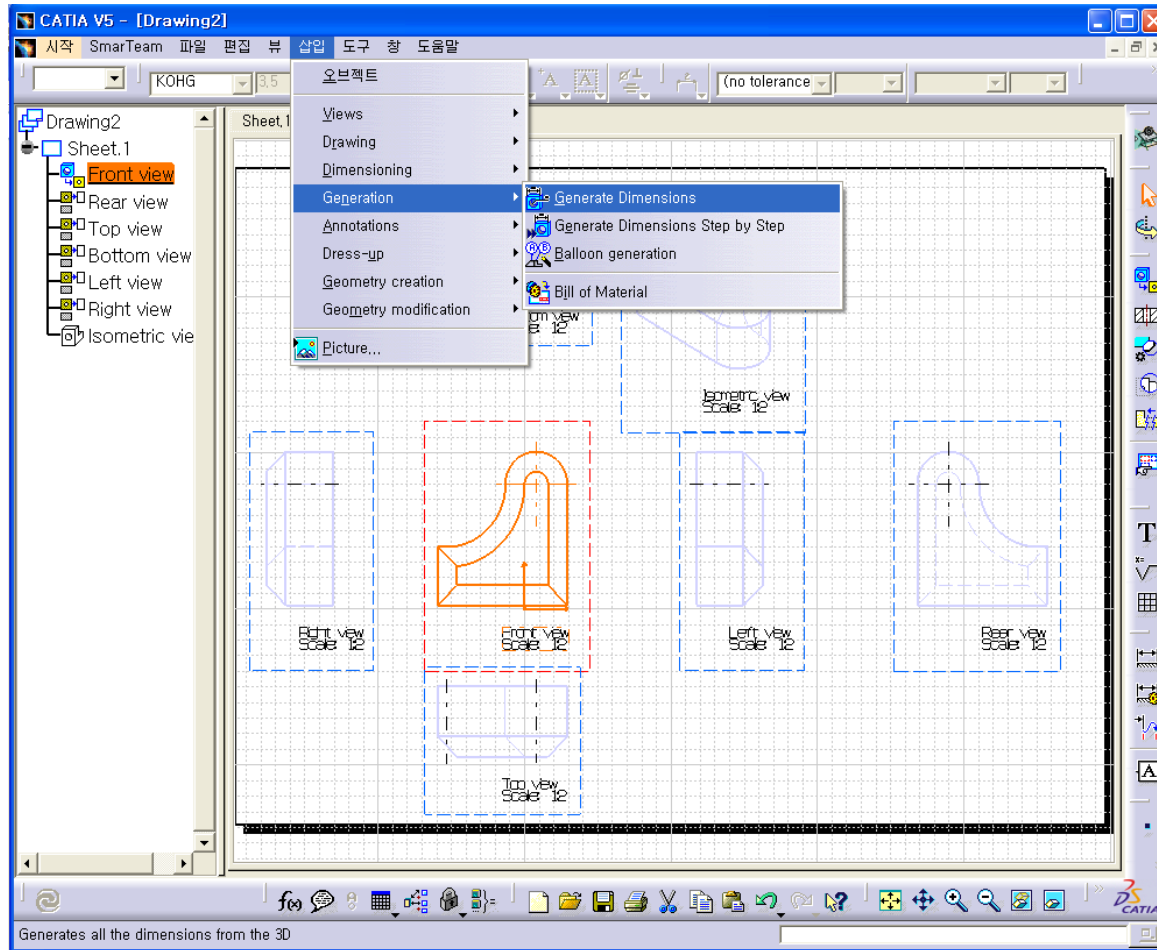
## 2. 단일 부품 드래프팅 (3)





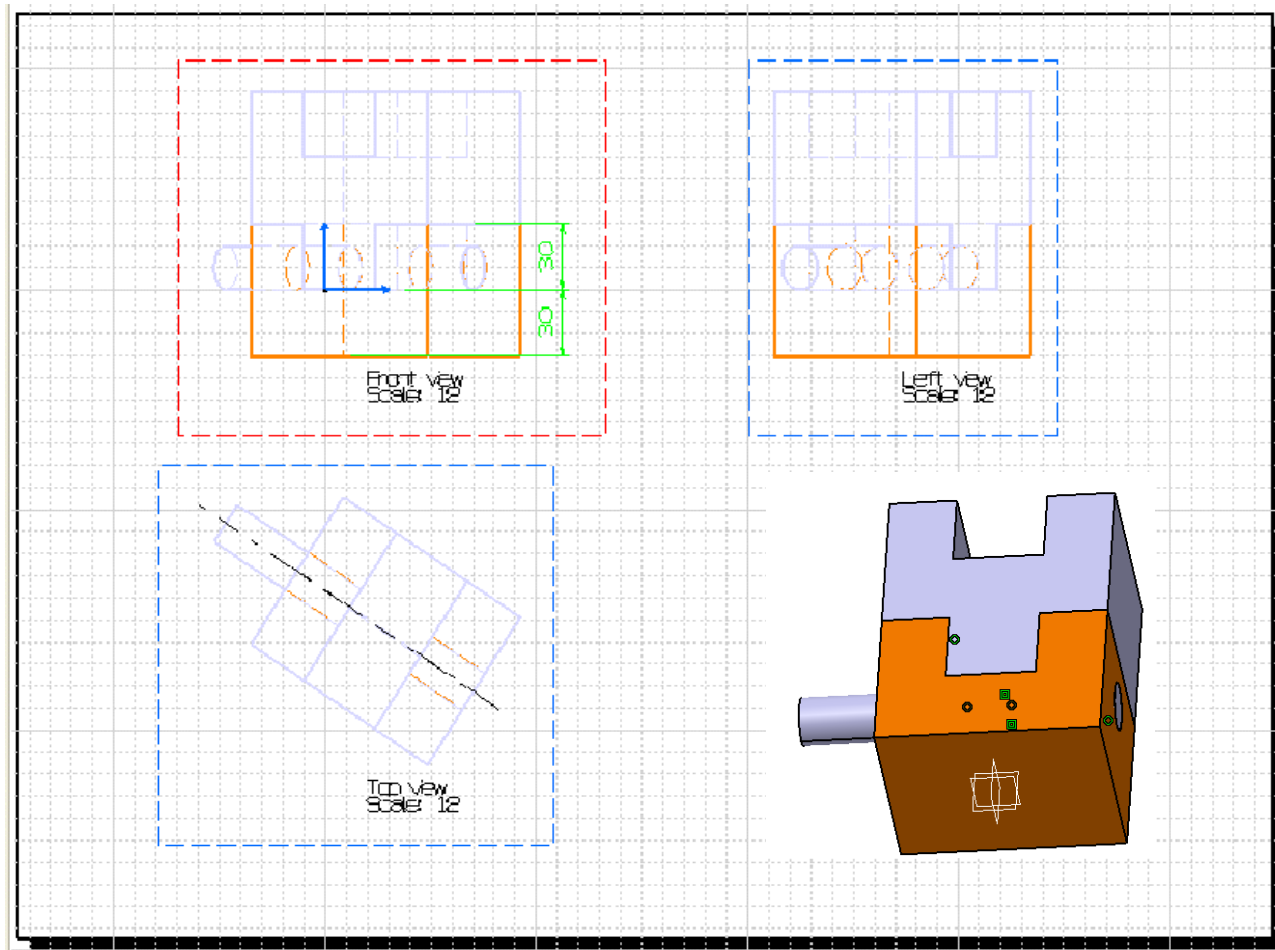
# VII. DRAFTING

## 2. 단일 부품 드래프팅 (4)



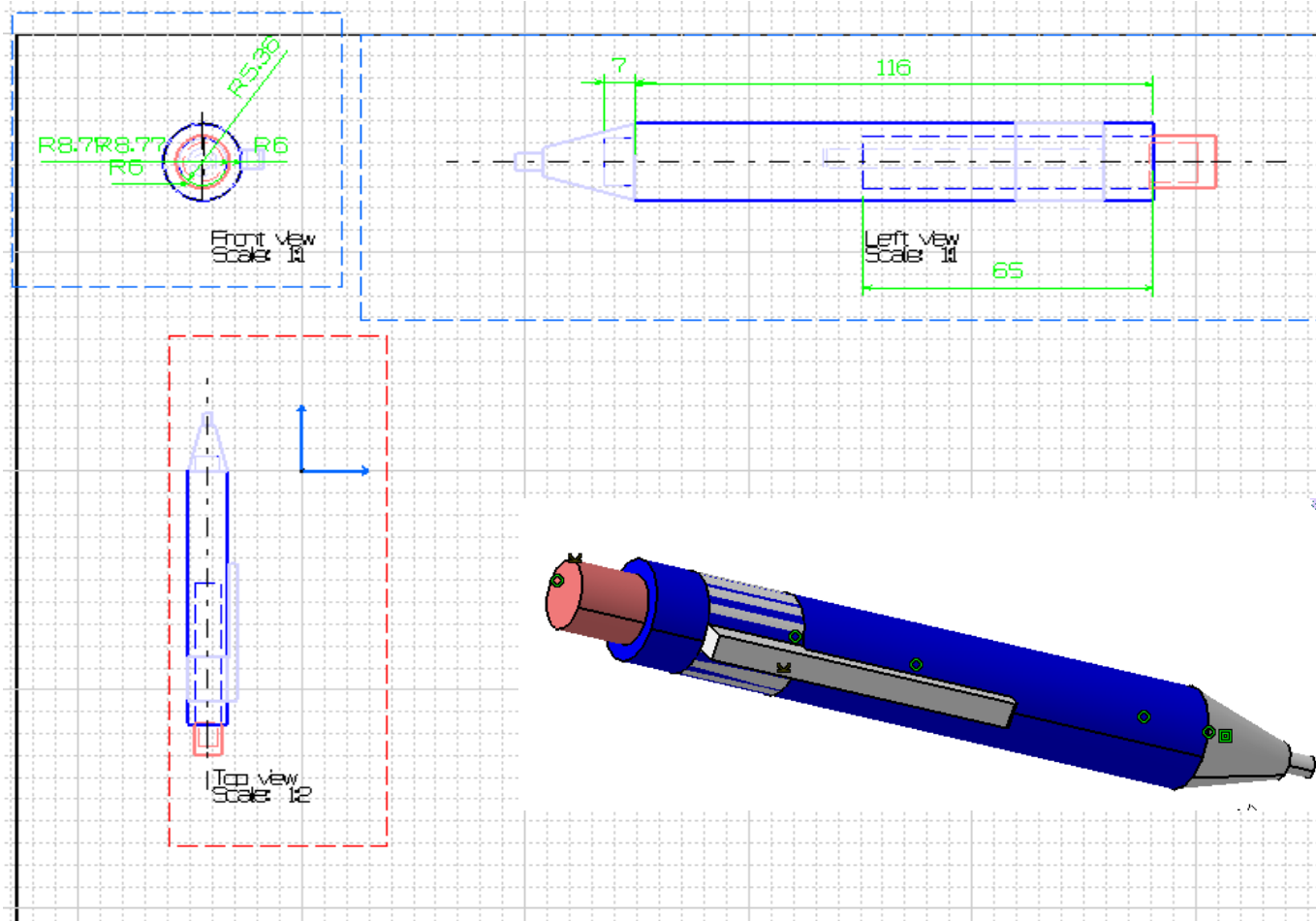
## VII. DRAFTING

### 3. 조립품 드래프팅 (1)



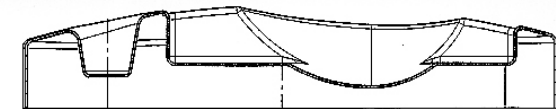
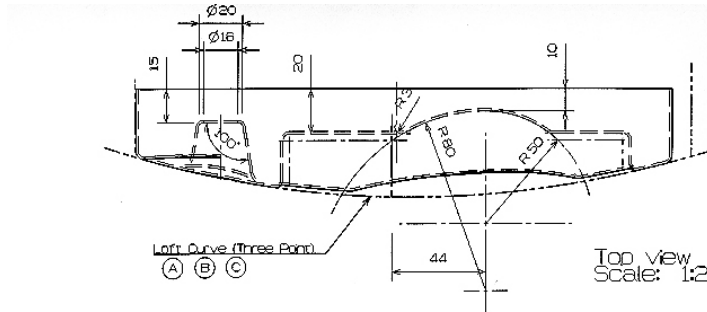
## VII. DRAFTING

### 3. 조립품 드래프팅 (2)



# VII. DRAFTING

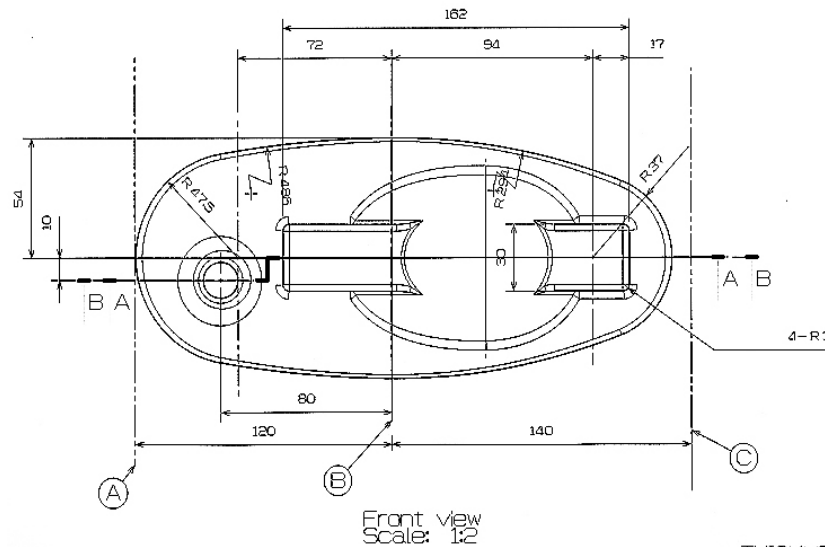
## 4. 드래프팅 예 제 (1)



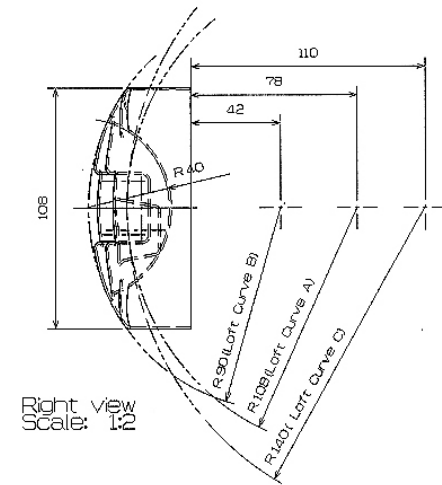
Section view A-A  
Scale: 1:2



Section cut B-B  
Scale: 1:2



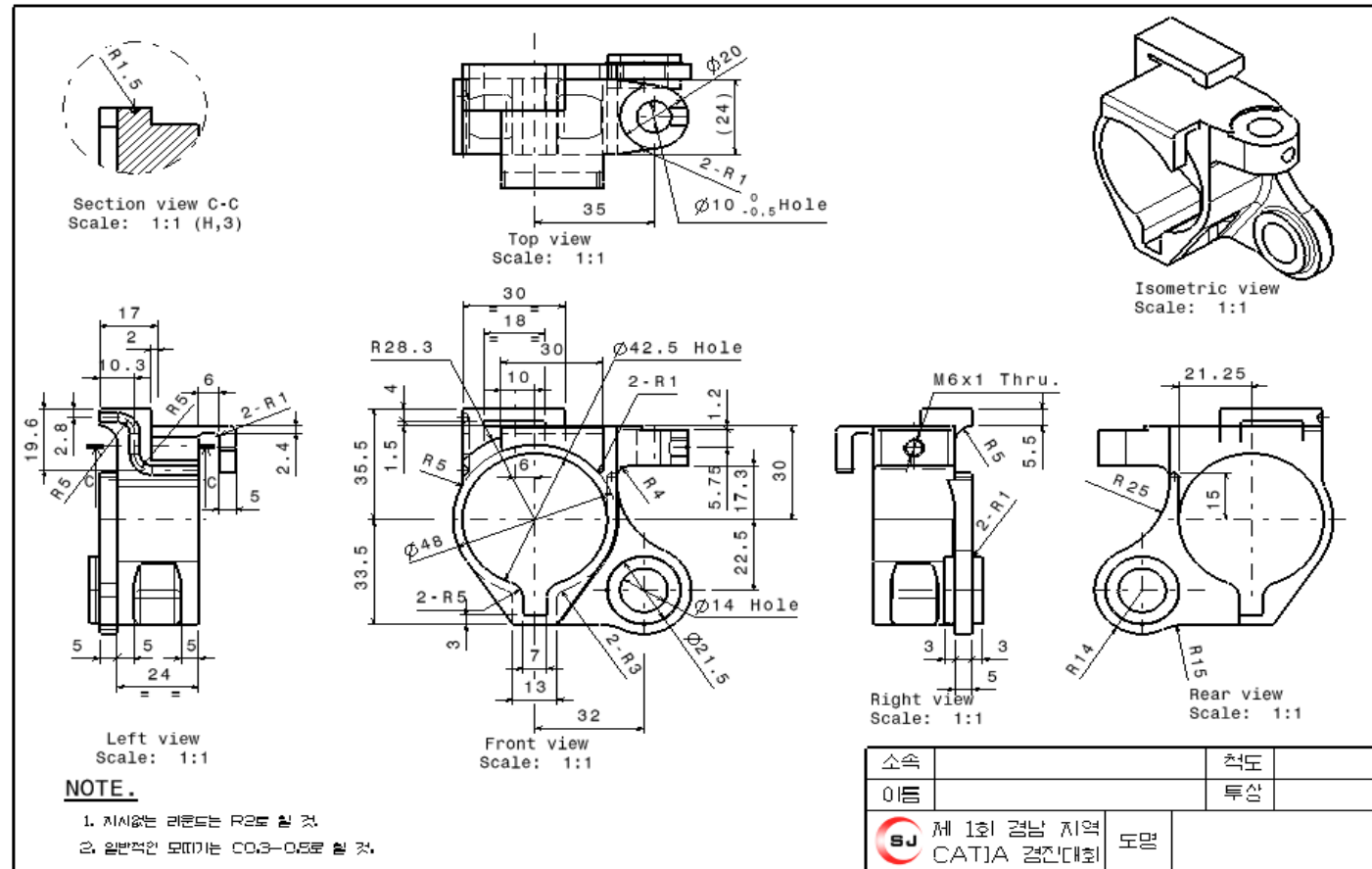
THICKNESS : 1mm



# VII. DRAFTING

## 4. 드래프팅 예 제 (2)

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