

CATIA V5 교안



2022. 03.

기계과 조 남 철 교수

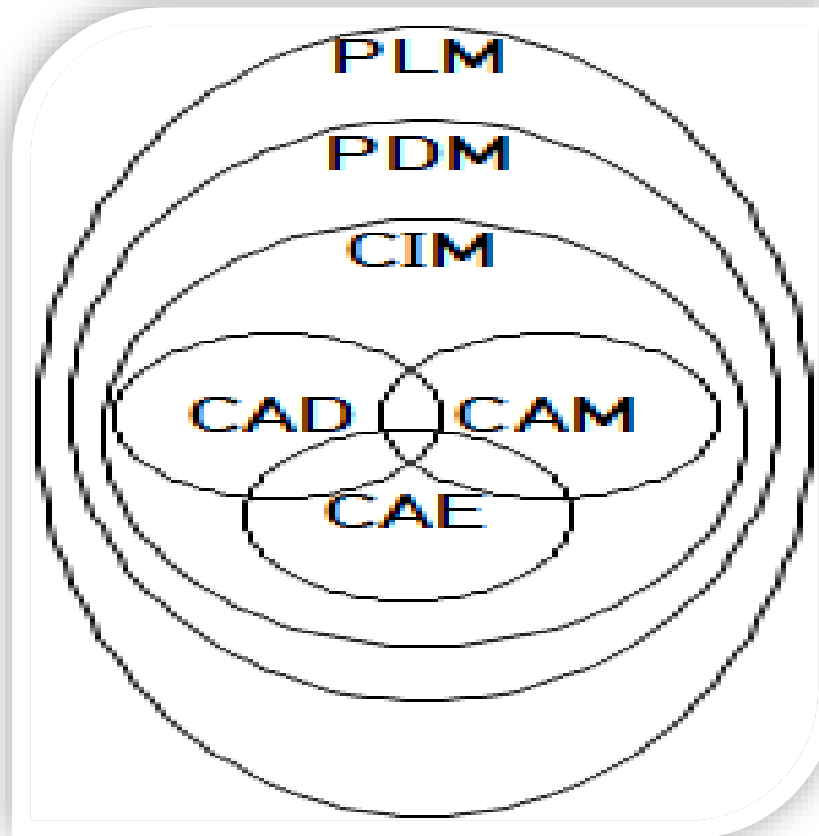
I. 3D CAD 의 개요

1. CAD 의 개요 및 관련 용어

CAD (Computer Aided Design & Drawing)	컴퓨터 응용 제품설계제도
CAM (Computer Aided Manufacturing)	컴퓨터 응용 제품제조생산
CAE (Computer Aided Engineering)	컴퓨터 응용 제품 해석 및 시뮬레이션
CIM (Computer Integrated Manufacturing)	CAD/CAM/CAE 의 통합 업무 시스템
FA (Factory Automation)	공장전체의 자동화
PDM (Computer Data Management)	제품 데이터 관리시스템
PLM (Computer Life-cycle Management)	제품 수명 전 기간 관리 시스템 (기획, 개념, 설계, 생산, 폐기 등)

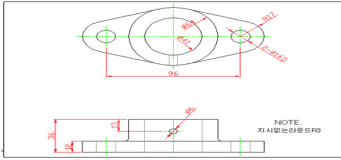
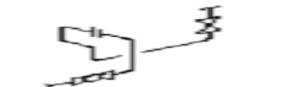

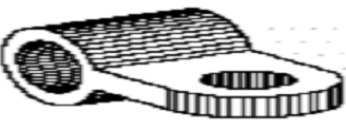
I. 3D CAD 의 개요

1. CAD 의 개요 및 관련 용어





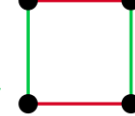
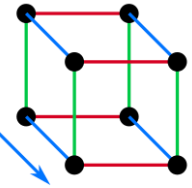
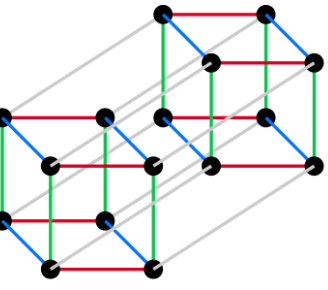

I. 3D CAD 의 개요

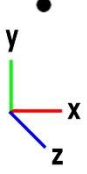

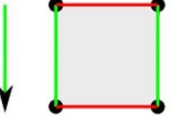
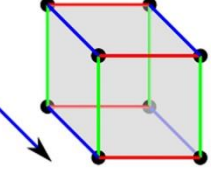
2. CAD 모델의 표현

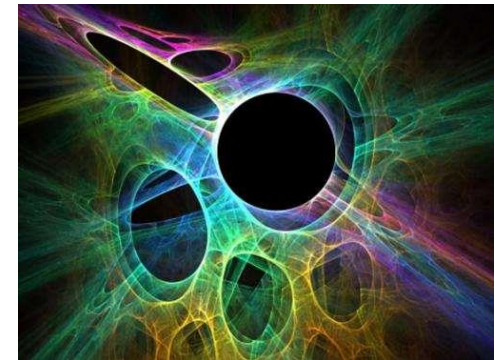
<p>2D Drafting</p>	 <p>NOTE: 지시된대로 50%</p>
<p>3차원 와이어 프레임 모델 (Wire-Frame Modeling)</p>	 <p>와이어프레임</p>
<p>서페이스 모델 (Surface Modeling)</p>	 <p>메쉬</p>
<p>솔리드 모델 (Solid Modeling)</p>	 <p>솔리드</p>

I. 3D CAD 의 개요

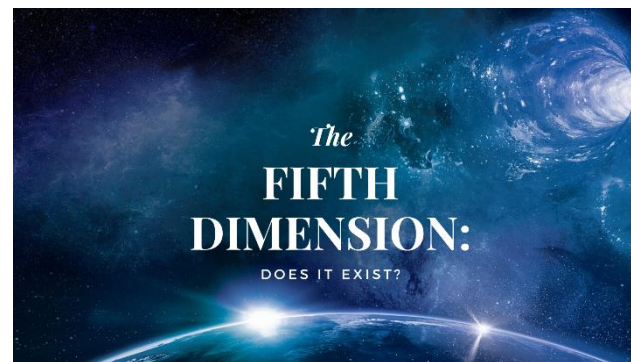
2. CAD 모델의 표현

					X Y Z W 
0	1	2	3	4	#Dim

0D Point 	1D Line 	2D Square 	3D Cube 
--	---	---	---

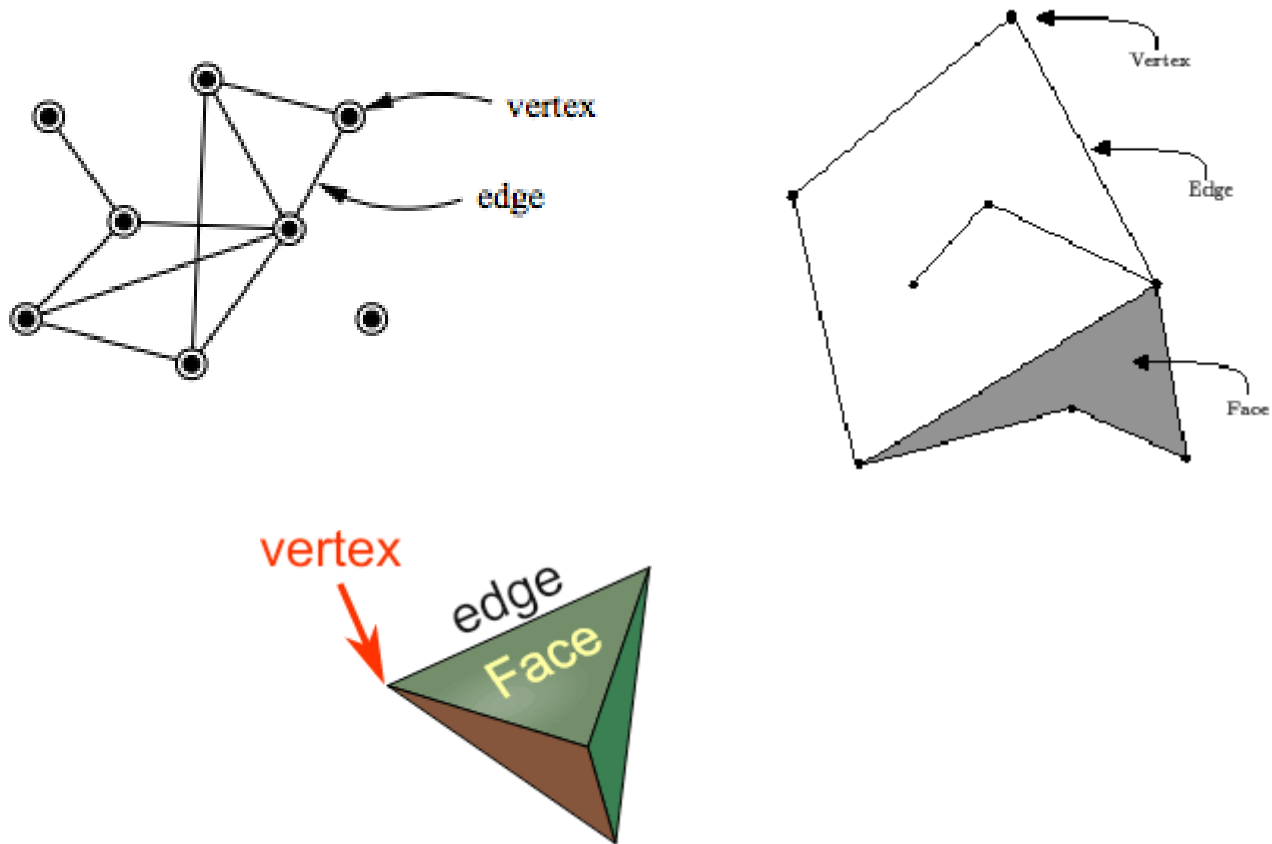


Superstring theory posits that the universe exists in 10 dimensions at once



I. 3D CAD 의 개요

2. CAD 모델의 표현

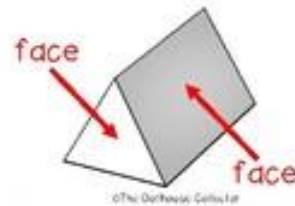


I. 3D CAD 의 개요

2. CAD 모델의 표현

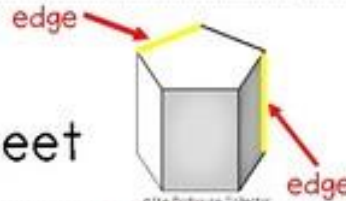
FACE

a flat surface of
a solid figure



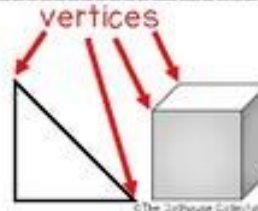
EDGE

where two faces meet



VERTEX




a point where two or more
line segments meet; a corner



3 D Shapes



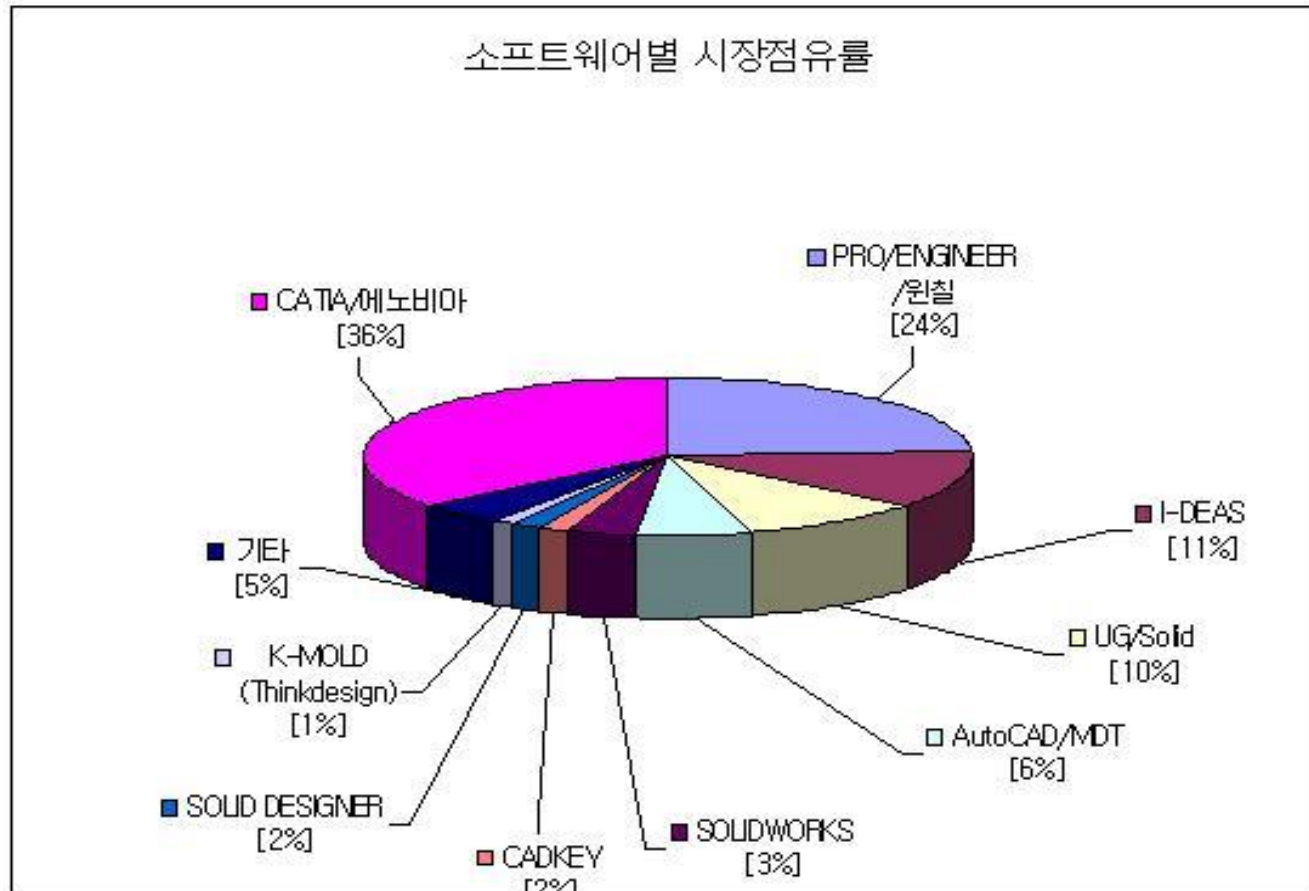
4 5 6 7 8 9 10 11 12 13 14 15

3 D SHAPE	# of Edges	# of Faces	# of Vertices
	12	6	8
	15	7	10
	9	5	6



I. 3D CAD 의 개요

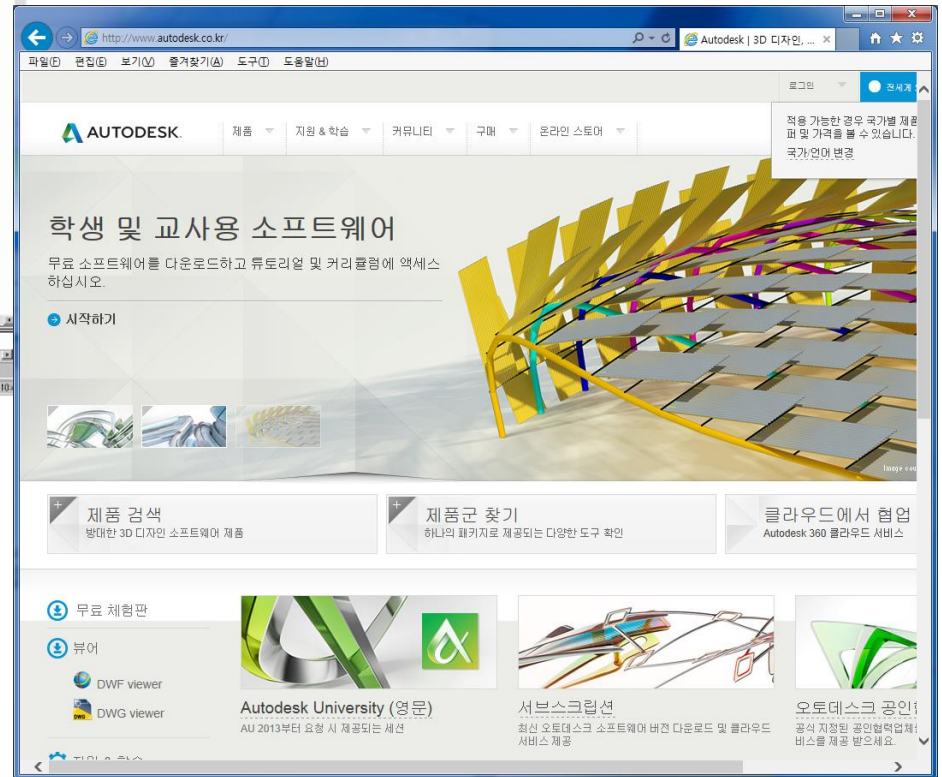
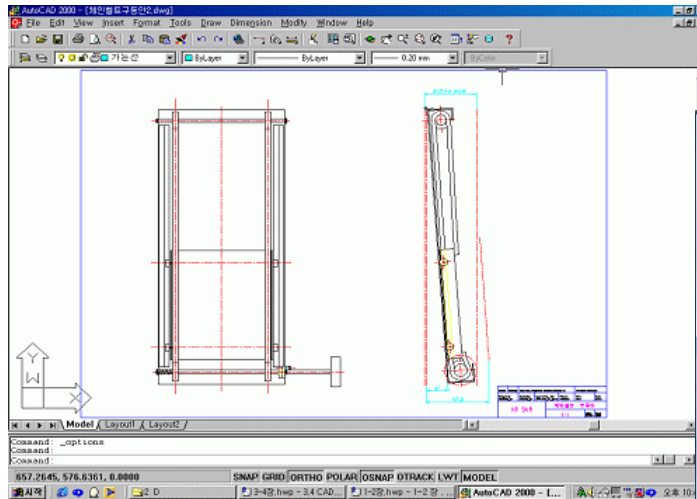
3. 각종 CAD Software



I. 3D CAD 의 개요

3-1. Auto CAD

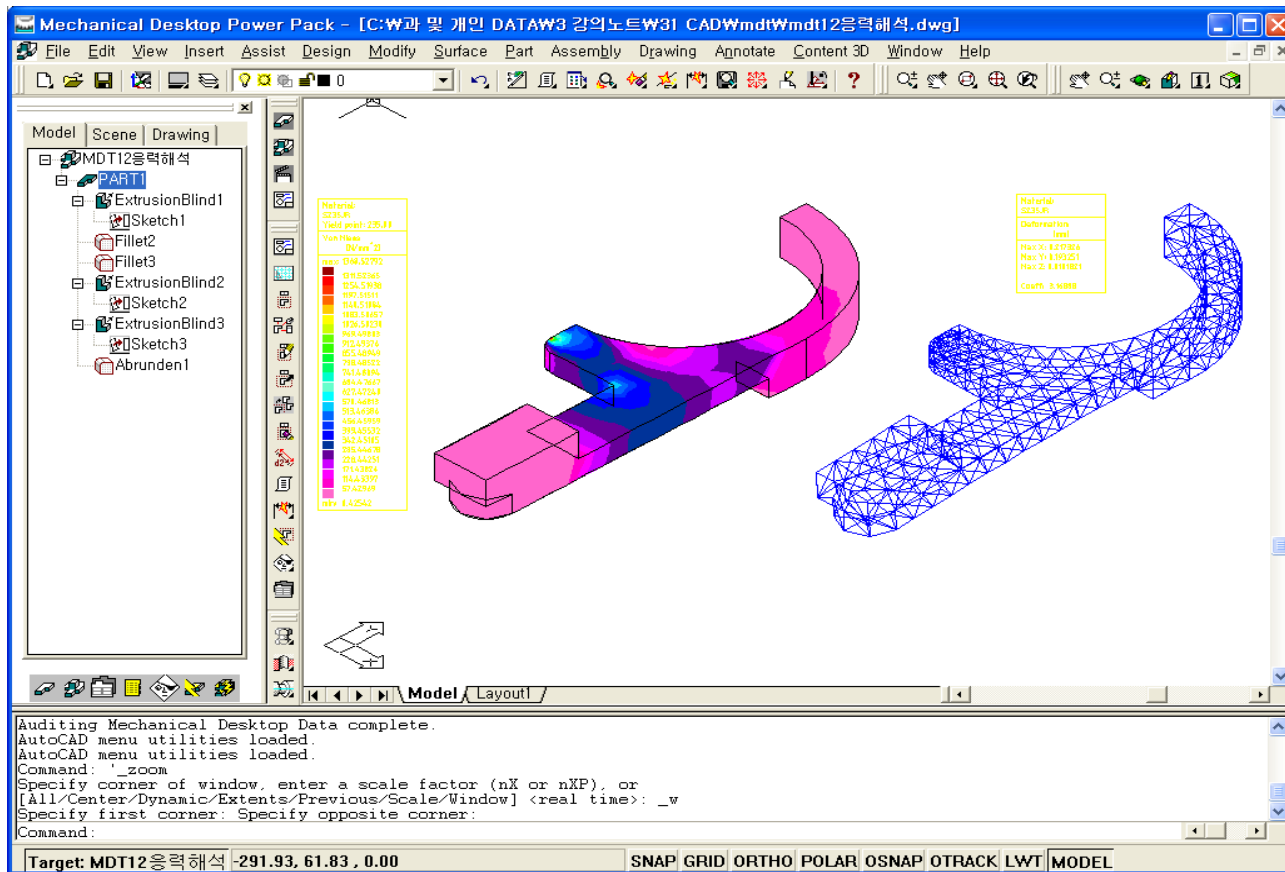
www.autodesk.co.kr



I. 3D CAD 의 개요

3-2. MDT (1)

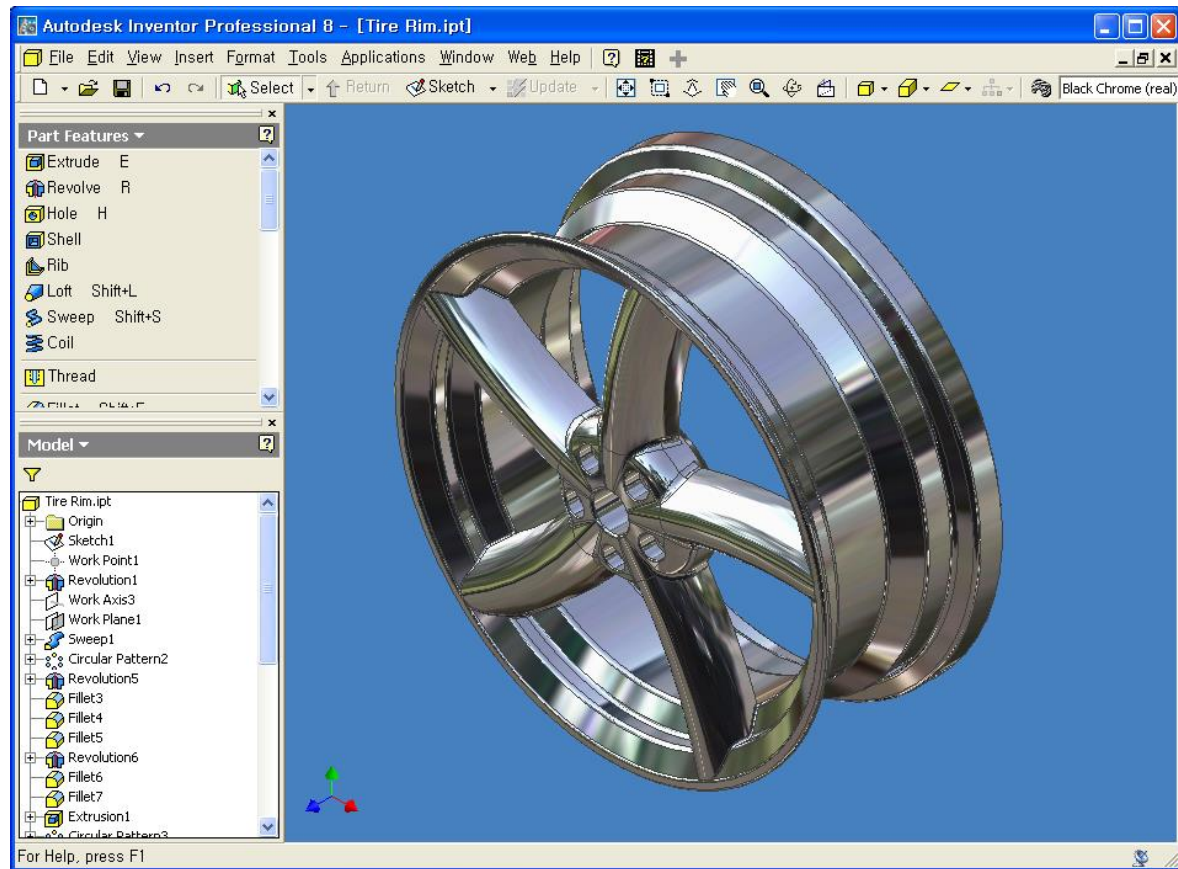
www.autodesk.com



I. 3D CAD 의 개요

3-2. Inventor (2)

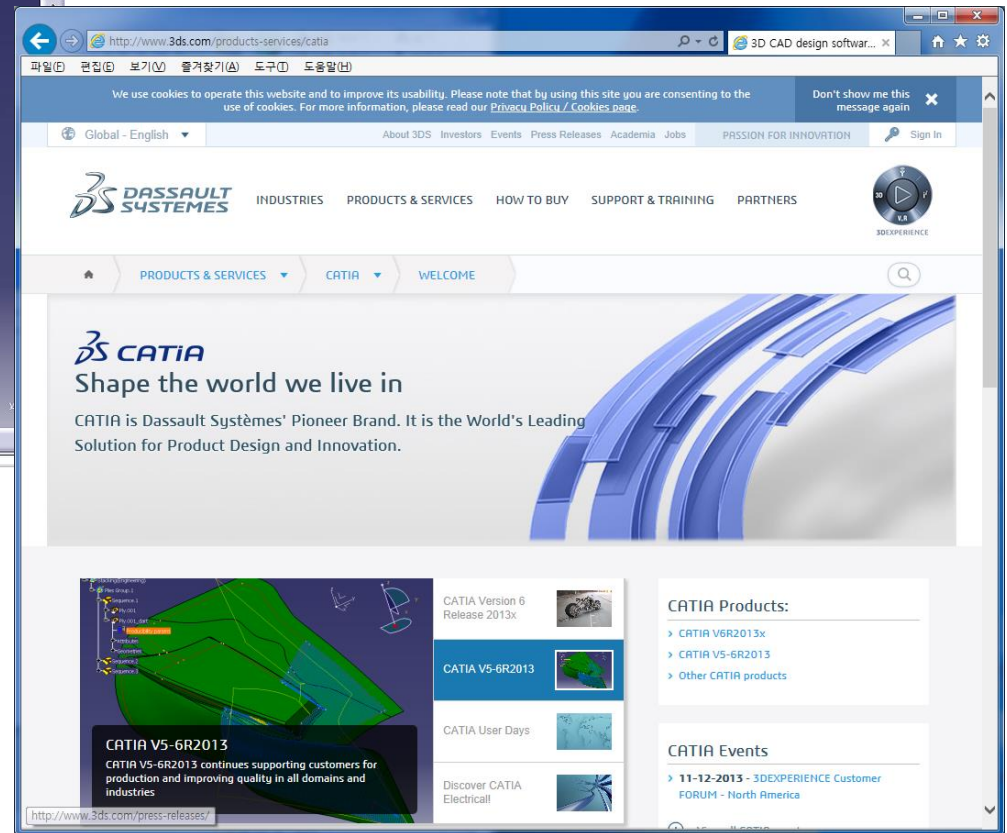
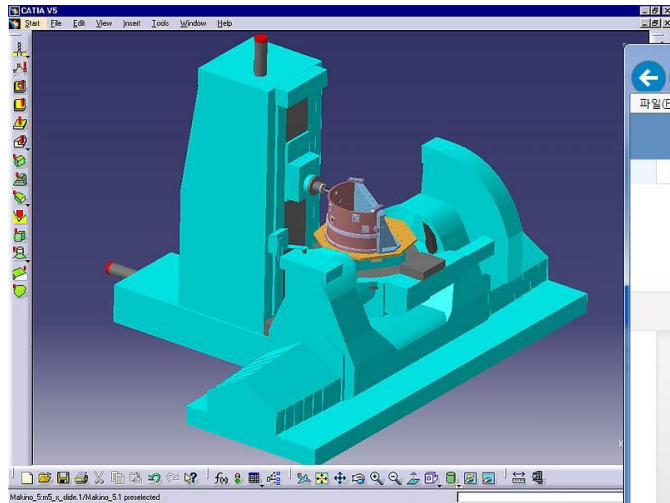
www.autodesk.com



I. 3D CAD 의 개요

3-3. CATIA

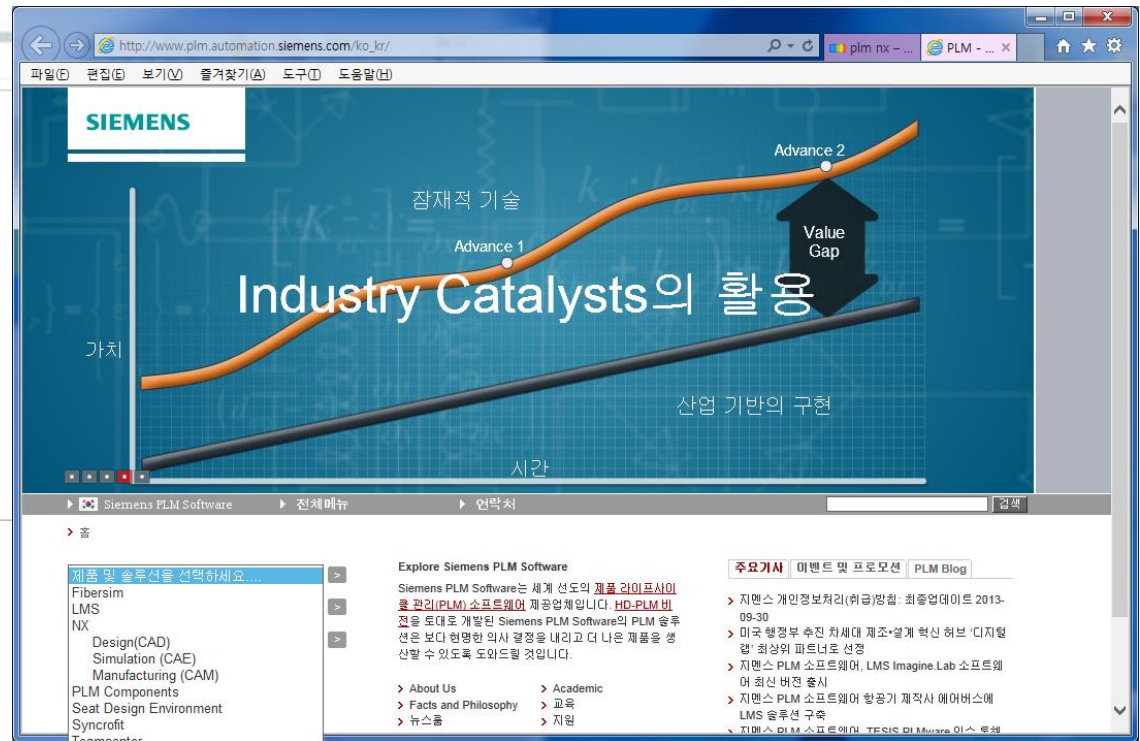
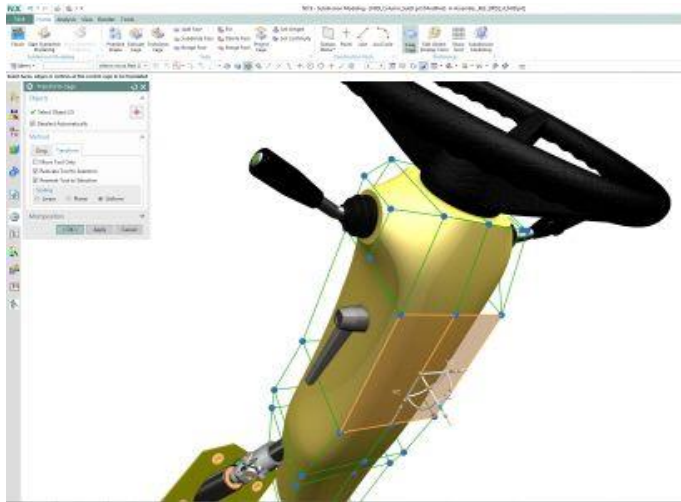
www. catia. com



I. 3D CAD 의 개요

3-4. NX, (UG-NX)

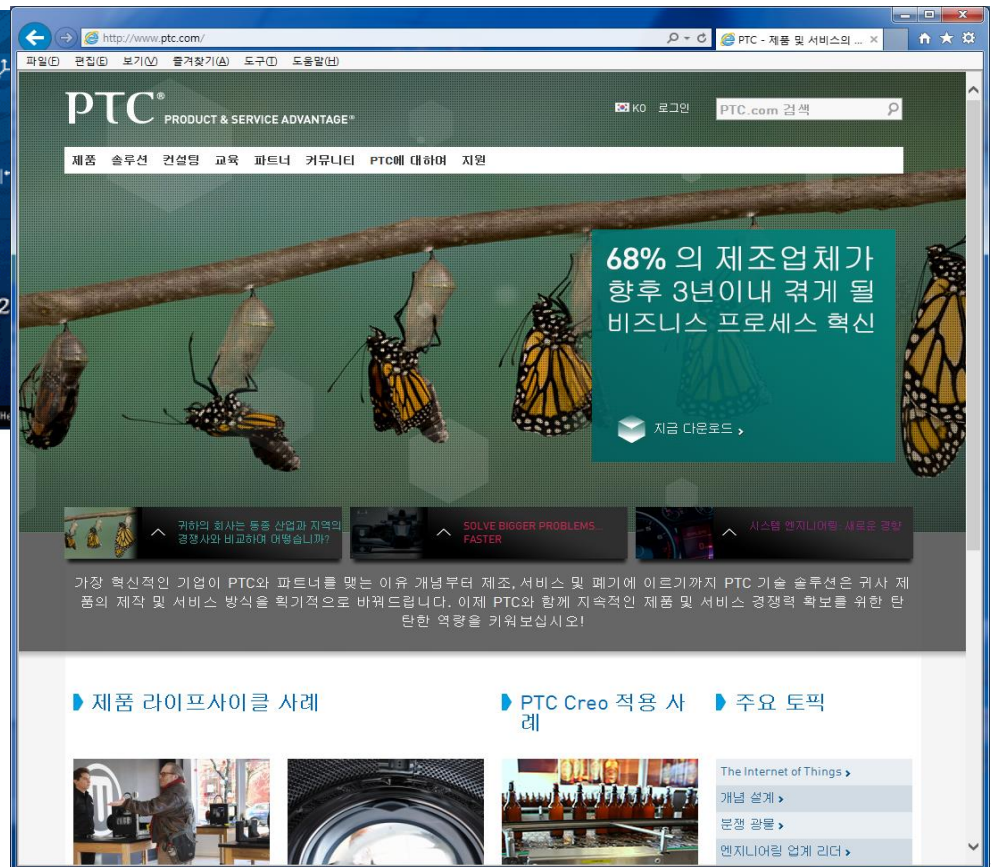
www.plm.automation.siemens.com



I. 3D CAD 의 개요

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

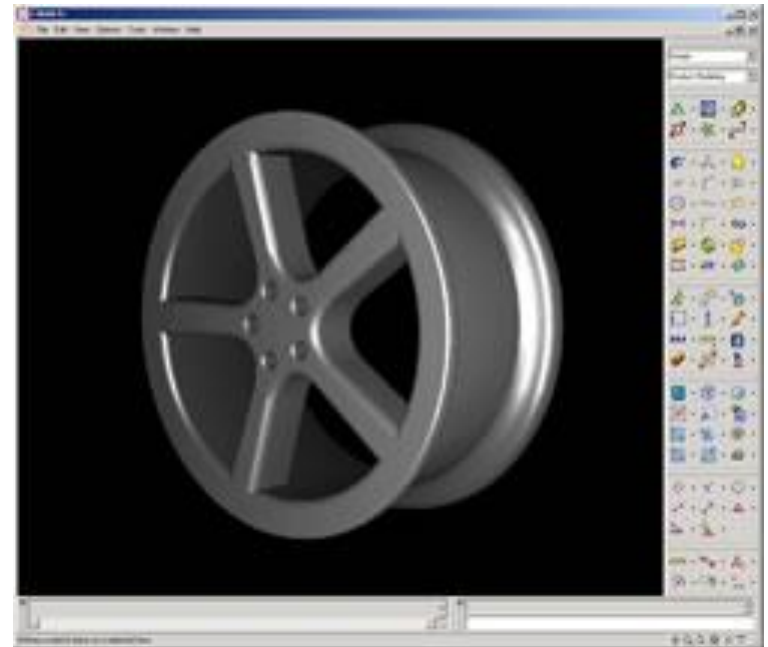
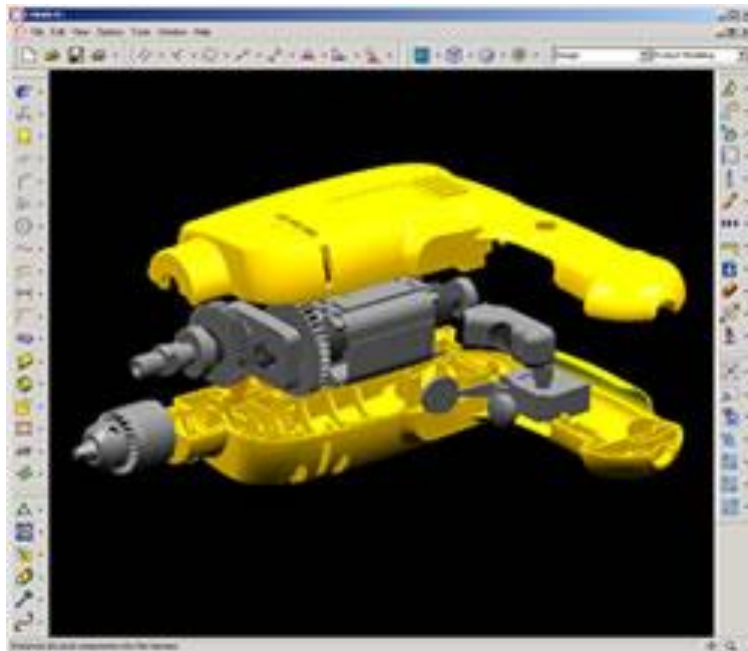
www.ptc.com



I. 3D CAD 의 개요

3-6. I-DEAS

지금은 지멘스 NX site (www.sdrc.com)



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

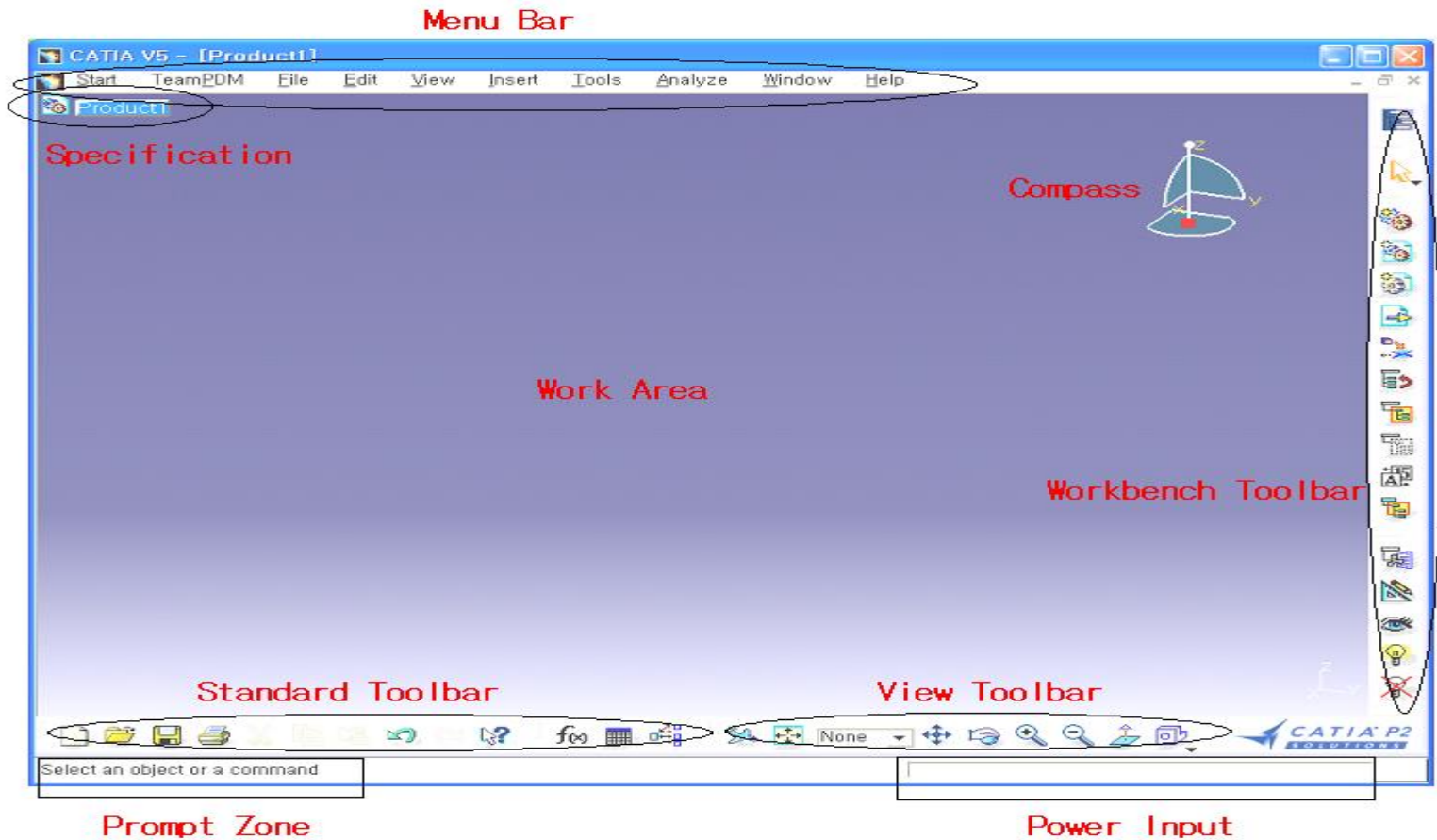
II. CATIA 의 개요

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

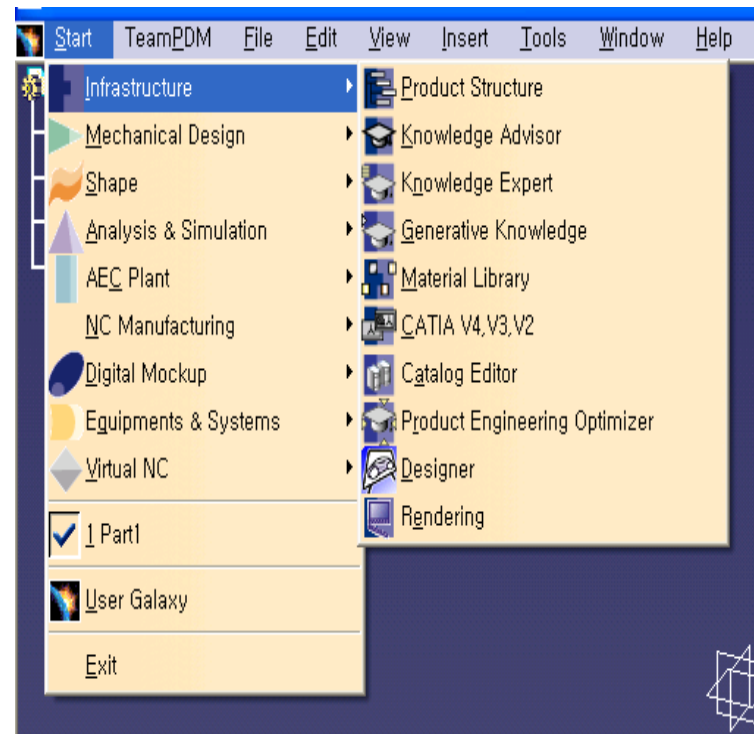
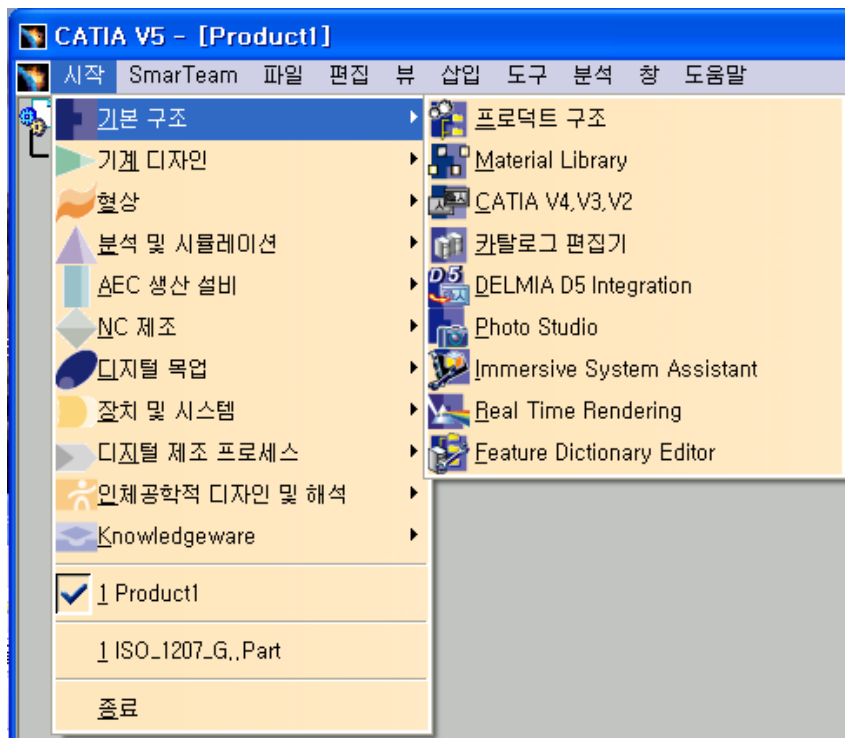
III. CATIA 의 구성 및 기초

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

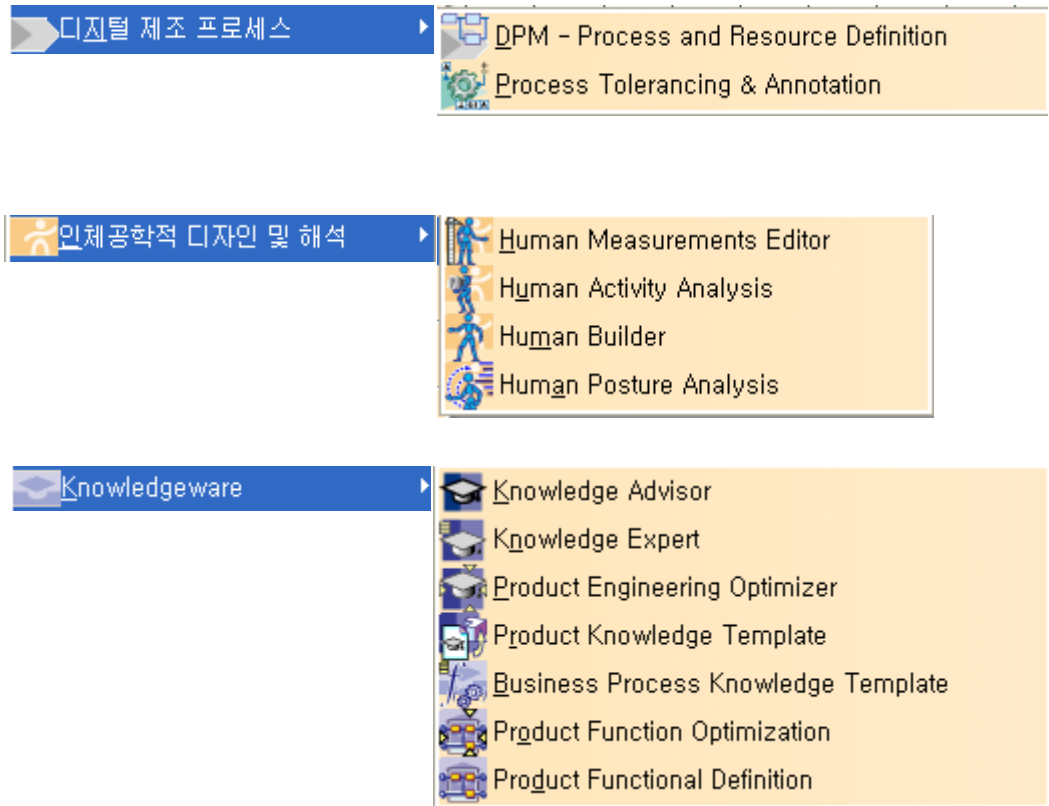
- Lathe Machining
- Prismatic Machining
- Surface Machining
- Advanced Machining
- NC Manufacturing Review
- STL Rapid Prototyping

디지털 목업

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

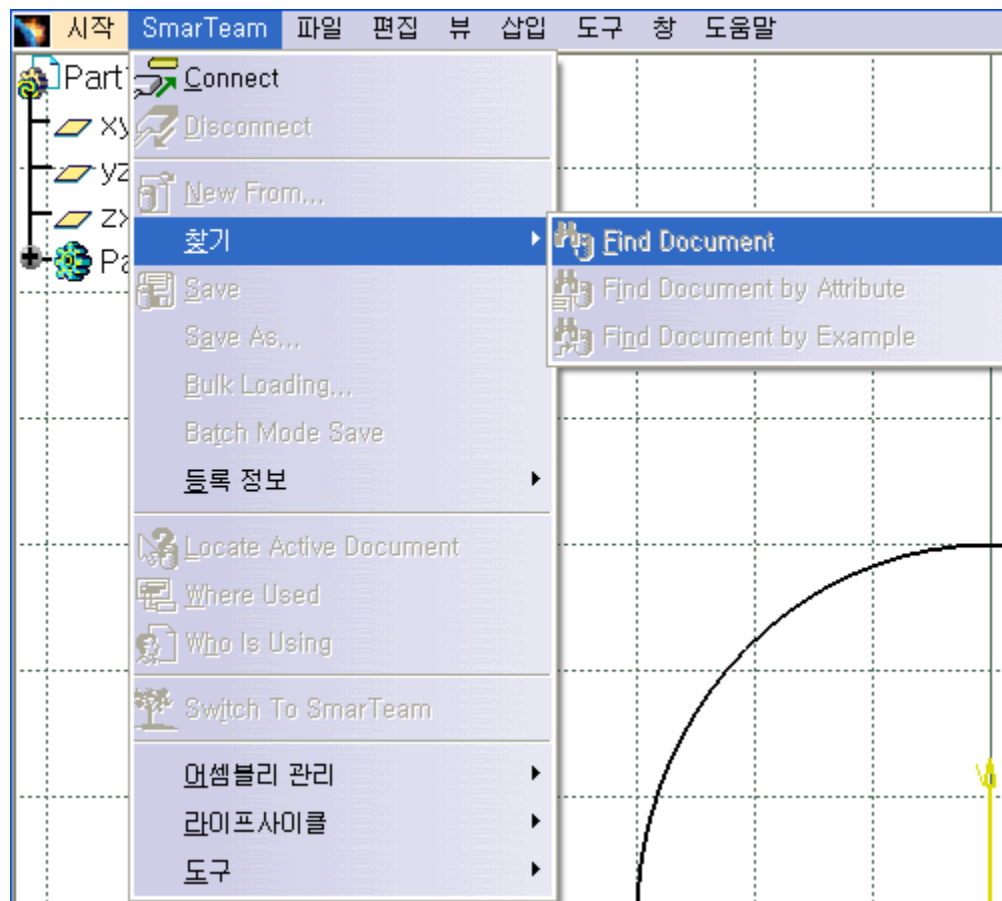
III. CATIA 의 구성 및 기초

2-1. 시작 (Start) (3)



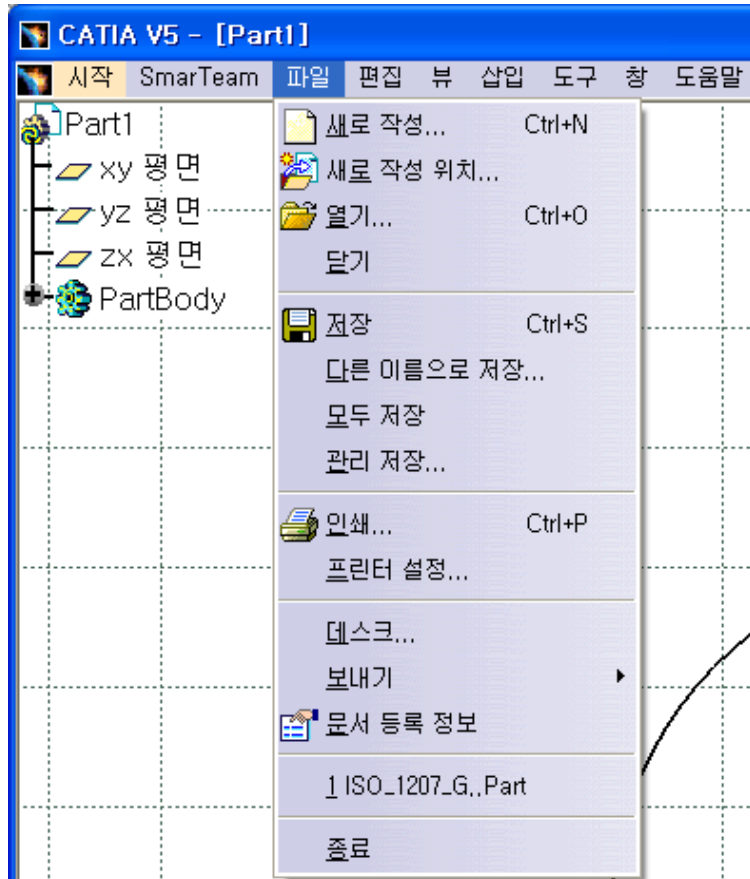
III. CATIA 의 구성 및 기초

2-2. 제품정보관리 (SmarTEAM)

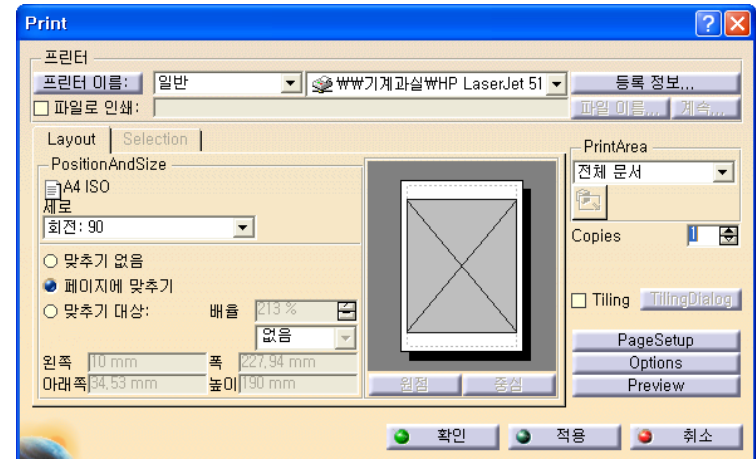
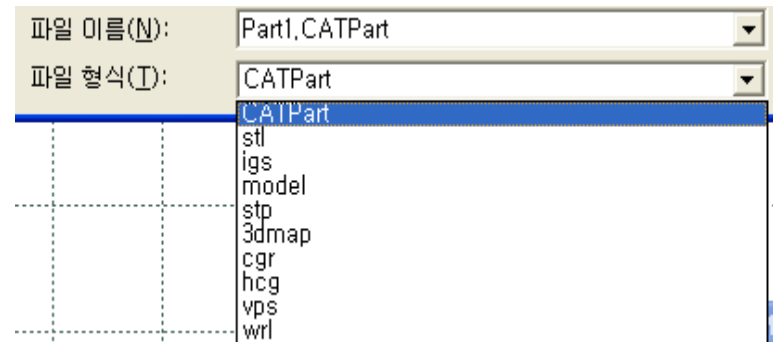


III. CATIA 의 구성 및 기초

2-3. 파일 (File)

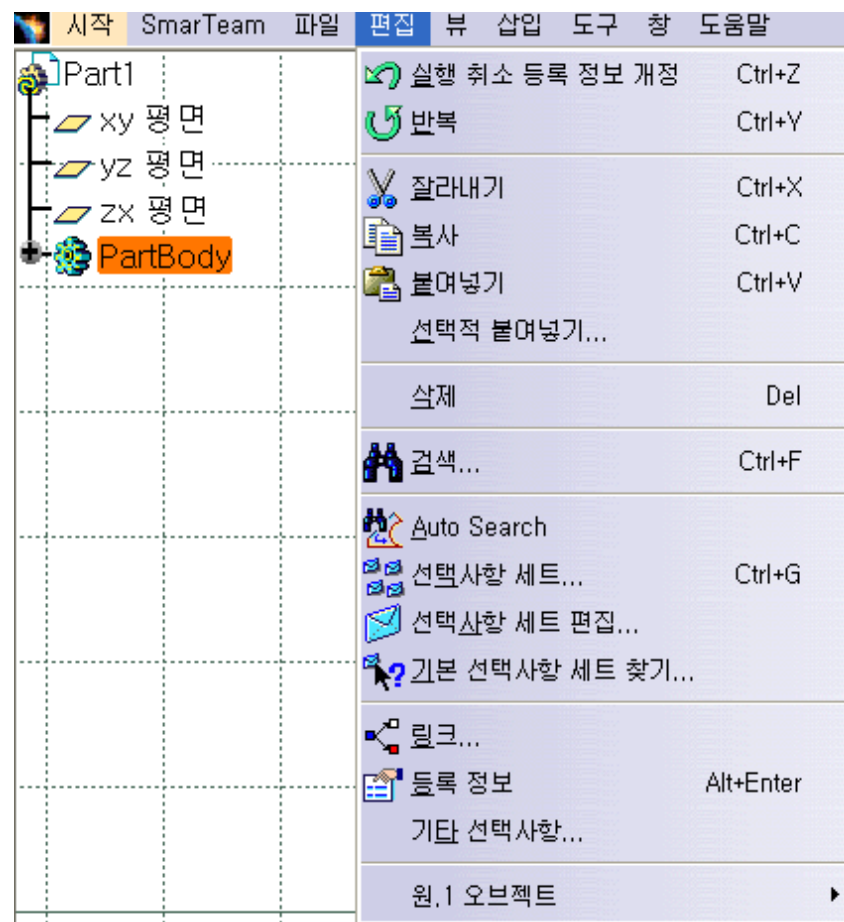


다른 이름으로 저장



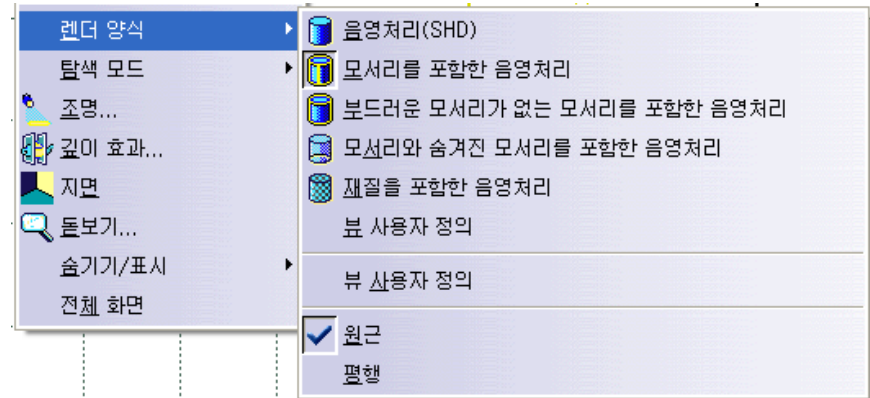
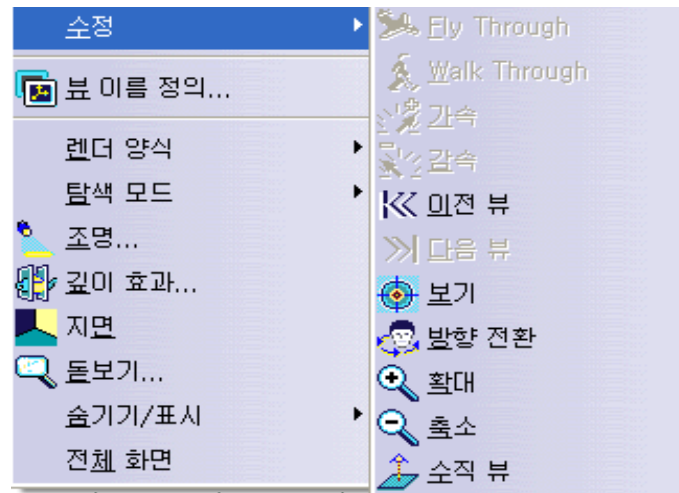
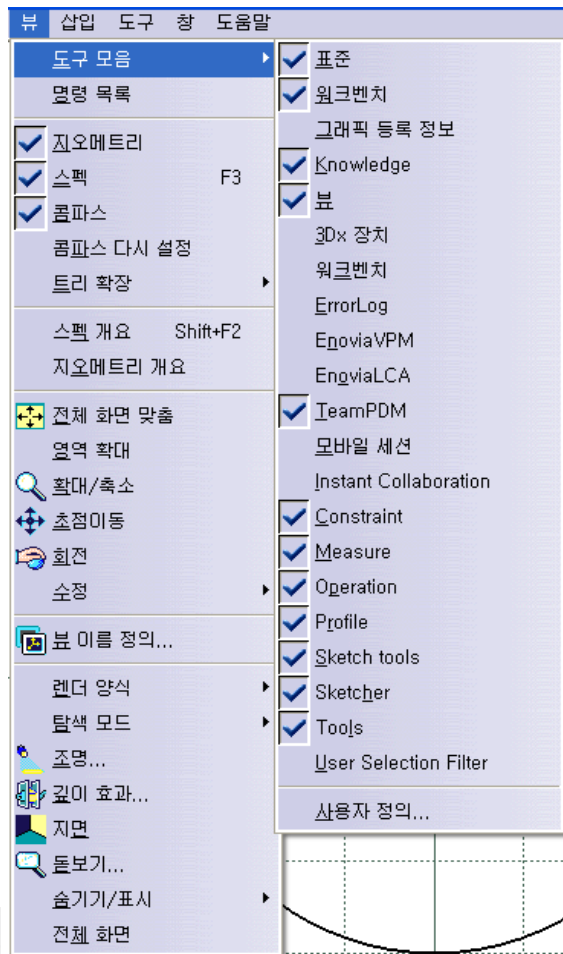
III. CATIA 의 구성 및 기초

2-4. 편집 (Edit)



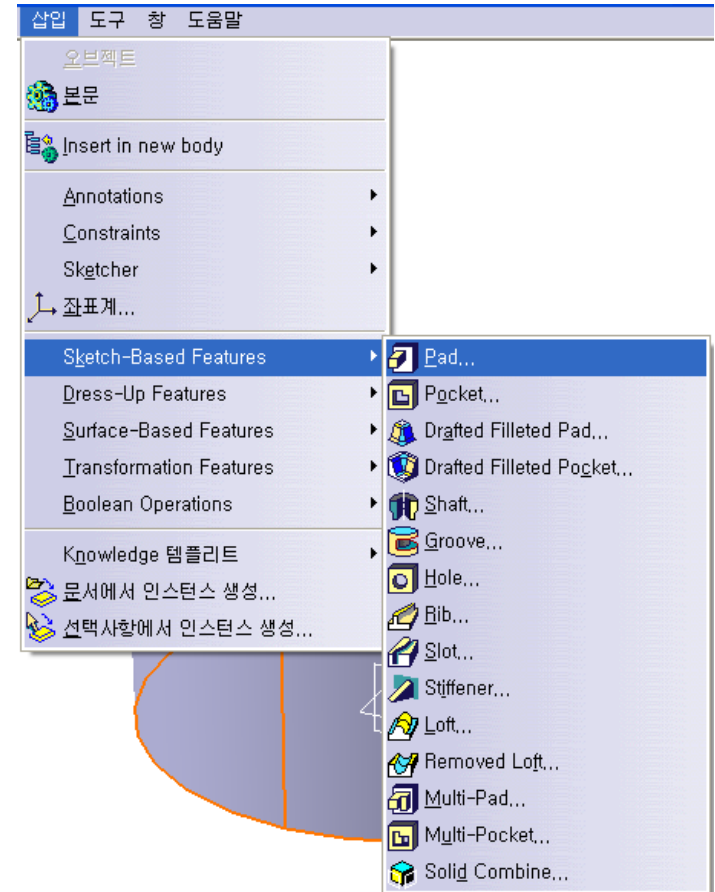
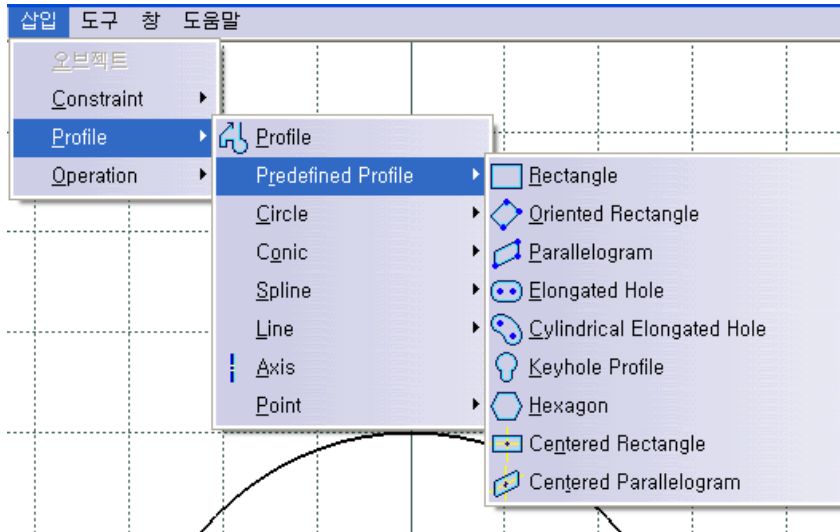
III. CATIA 의 구성 및 기초

2-5. 뷰 (View)



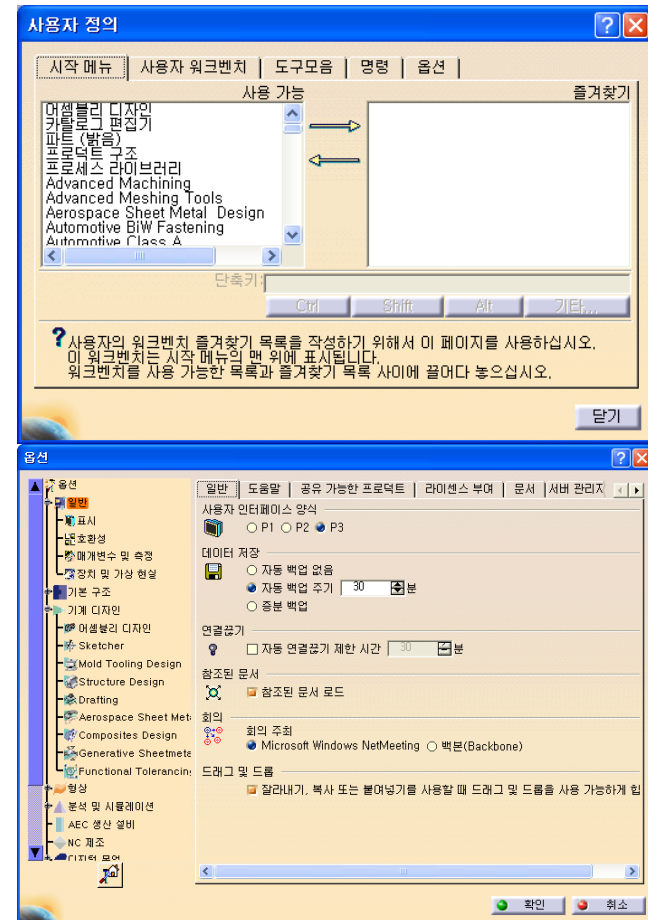
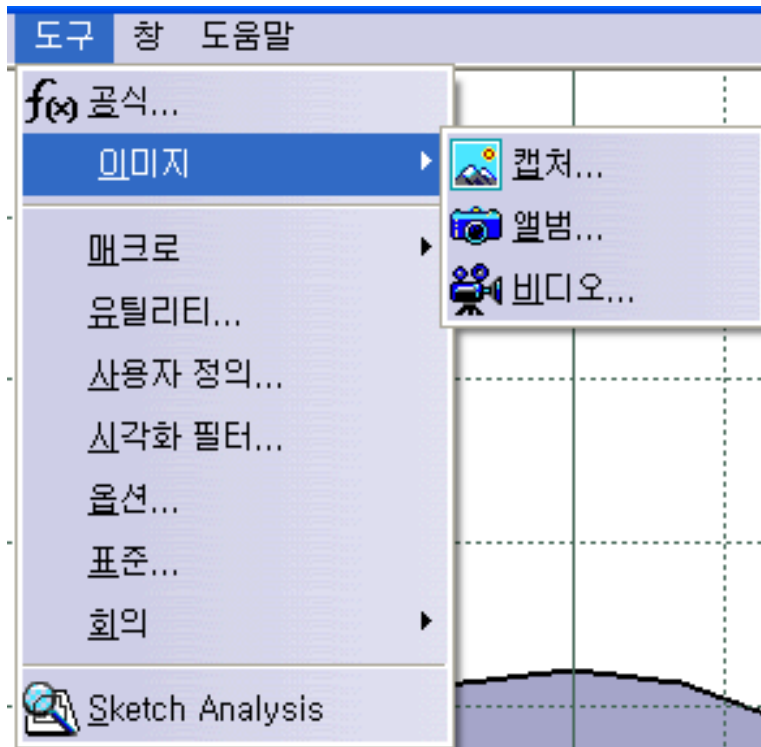
III. CATIA 의 구성 및 기초

2-6. 삽입 (Insert)



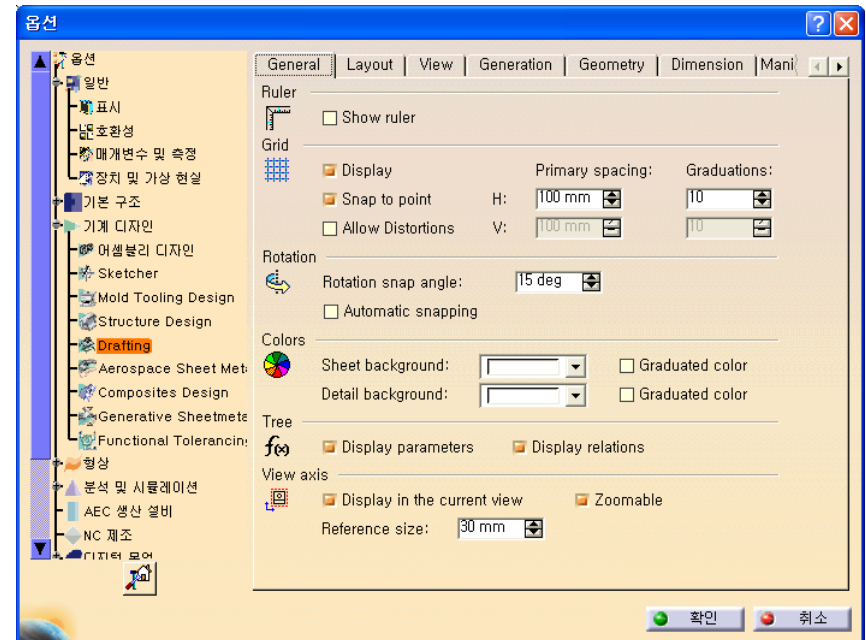
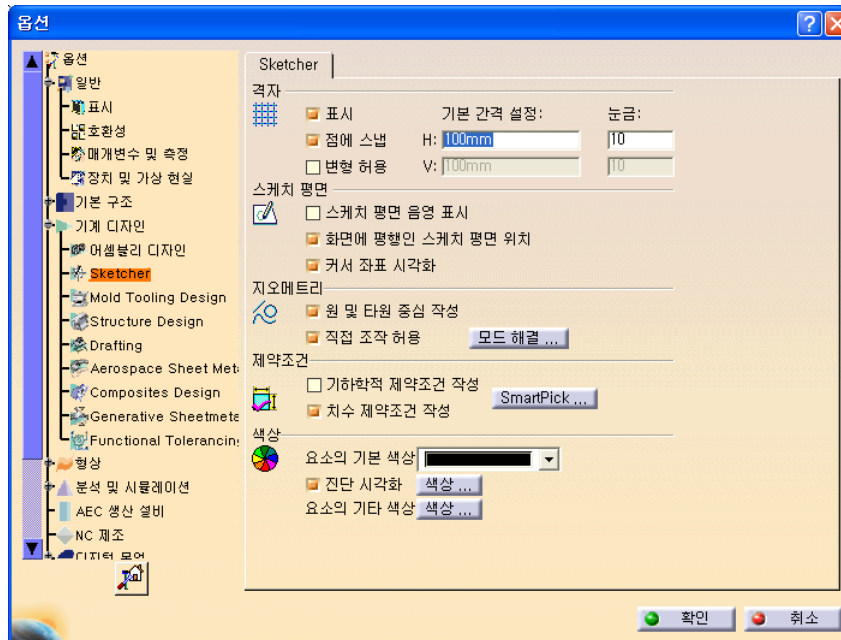
III. CATIA 의 구성 및 기초

2-7. 도구 (Tools)



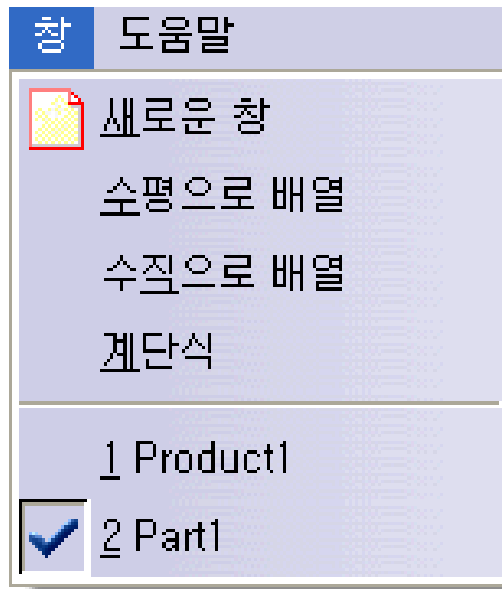
III. CATIA 의 구성 및 기초

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




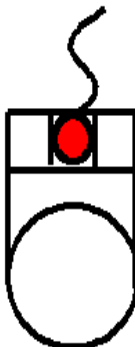

III. CATIA 의 구성 및 기초

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	사용 방법
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 뎀 → Drag

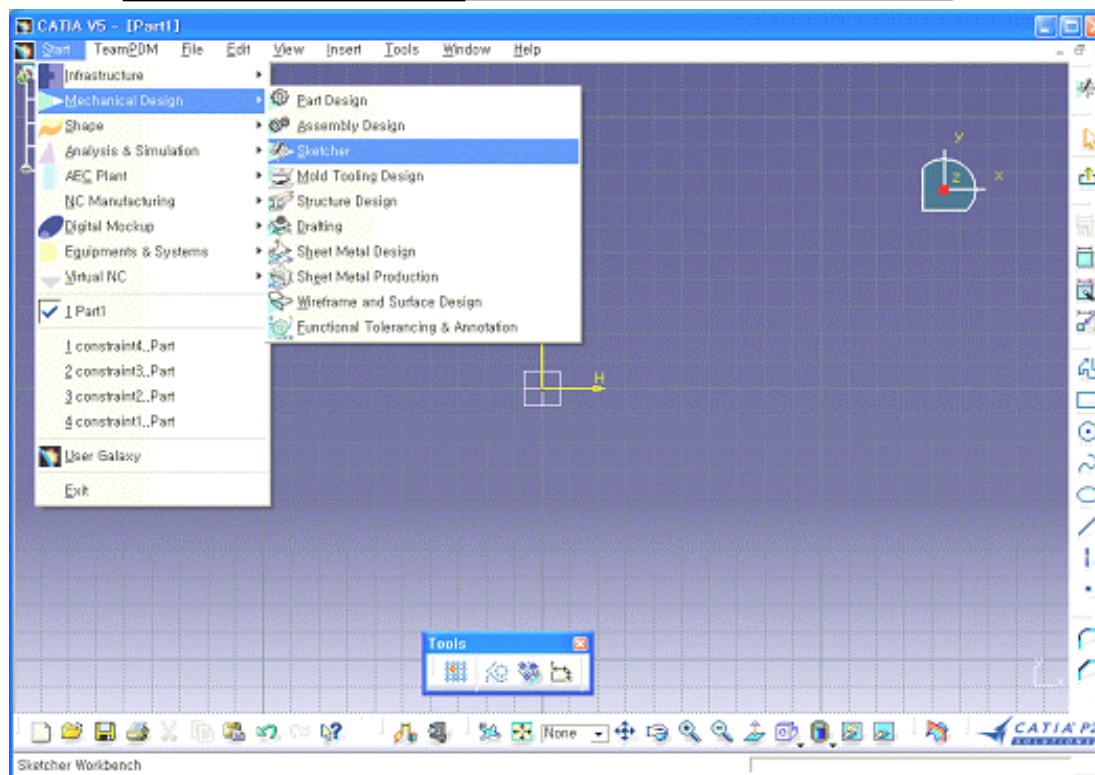
III. CATIA 의 구성 및 기초

4. CATIA 의 저장 파일

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

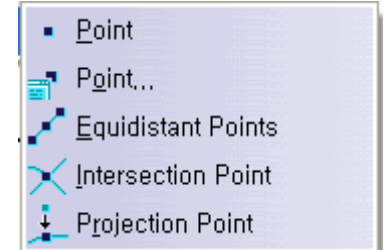
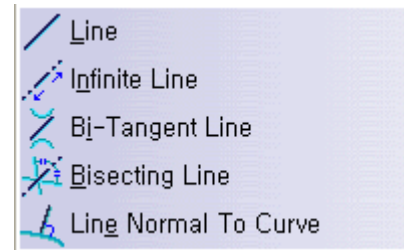
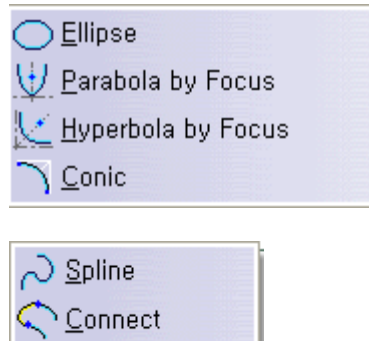
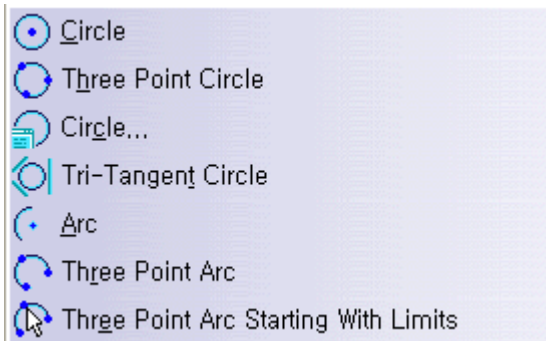
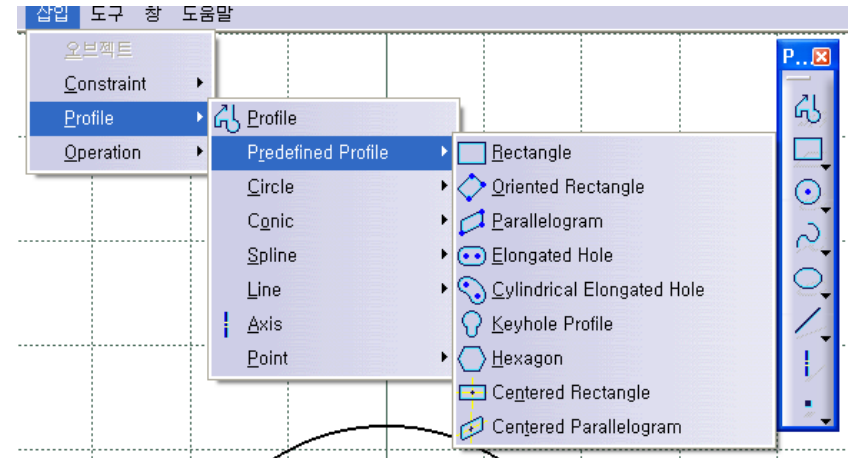
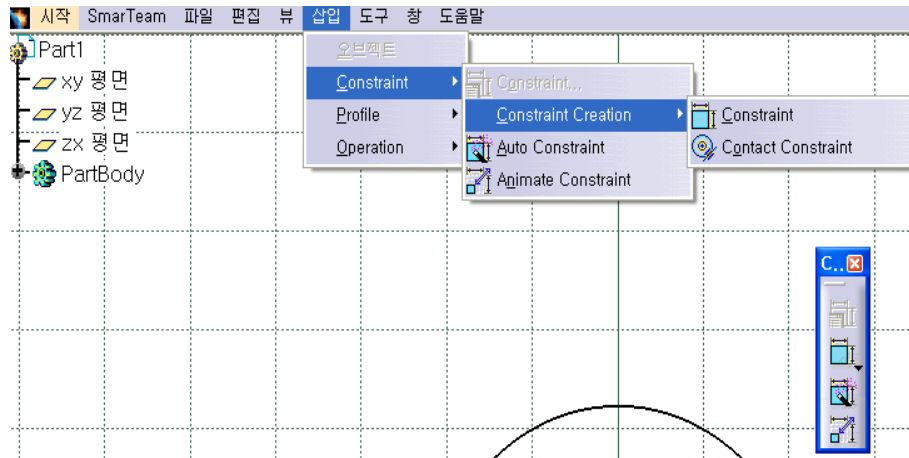
IV. SKETCHER

1. Sketcher 의 소개



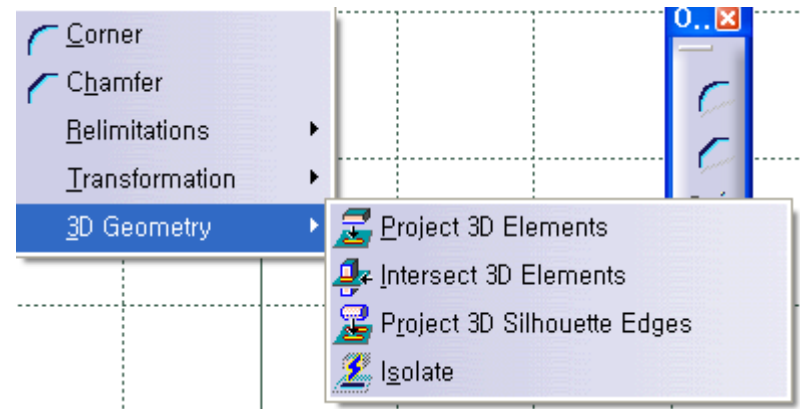
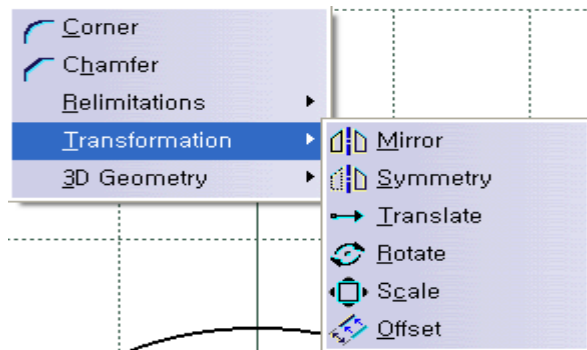
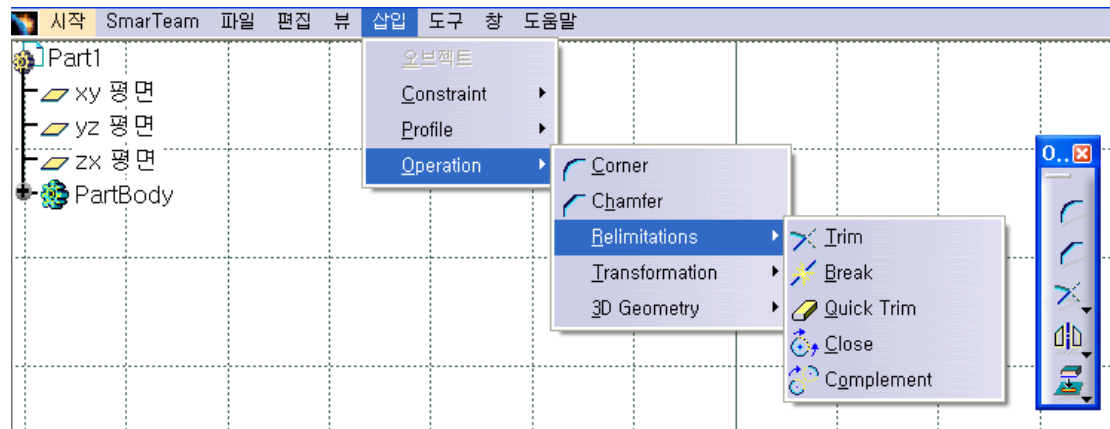
IV. SKETCHER

2. Sketcher 의 Insert 명령과 Icon (1)



IV. SKETCHER

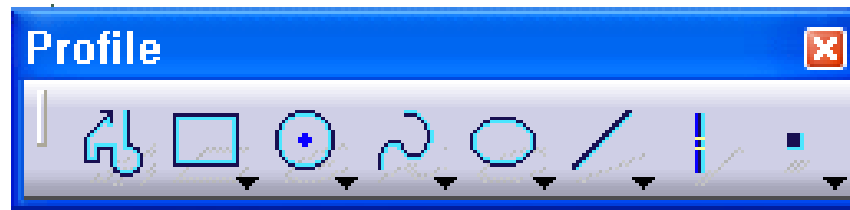
2. Sketcher 의 Insert 명령과 Icon (2)



IV. SKETCHER


3. Sketching Simple Profile, Profile

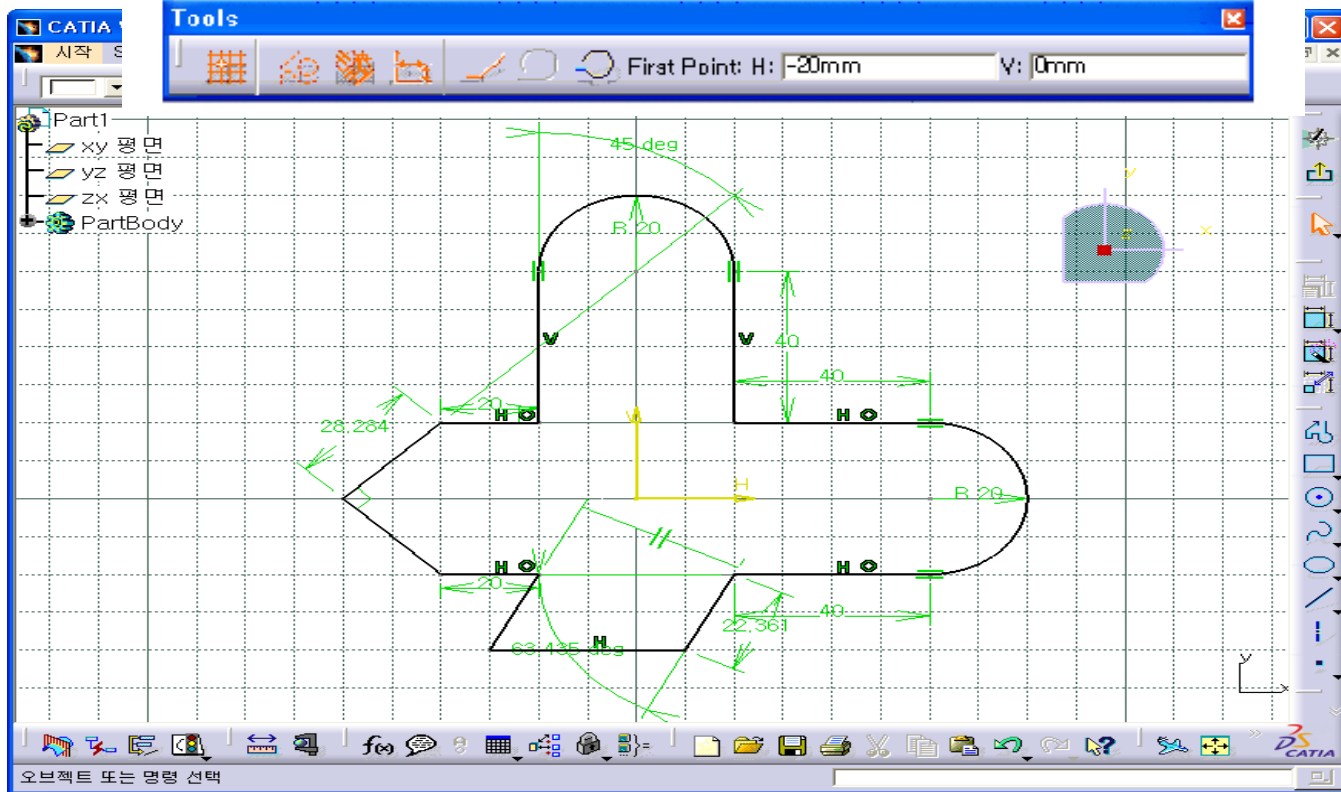
Sketcher 에서 Profile을 생성하는 가장 기본적인 툴 바다.



IV. SKETCHER

3-1. Profile

	PROFILE	Arc, Line, Circle을 혼합하여 생성
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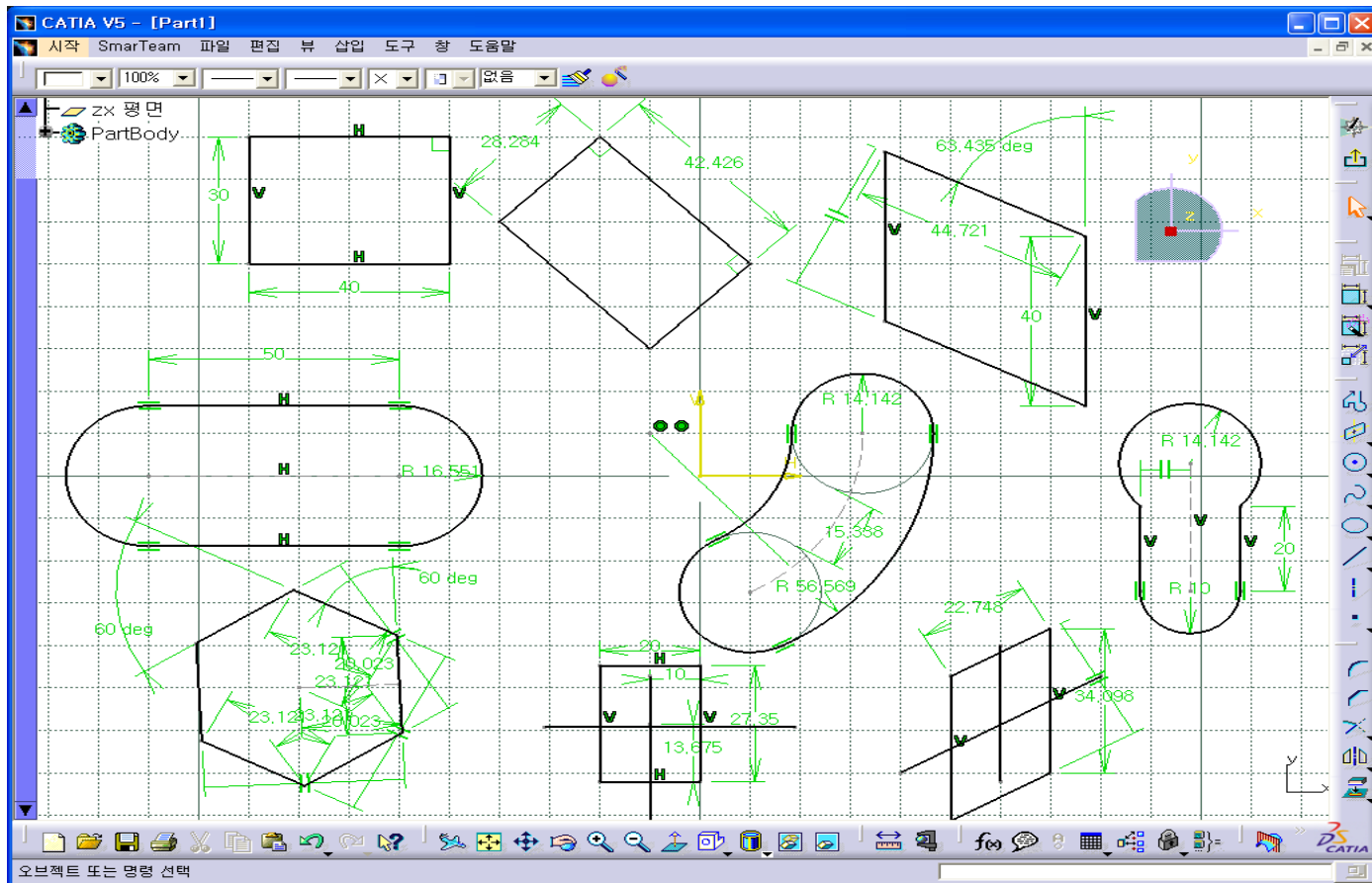
IV. SKETCHER

3-2. Rectangle (1)



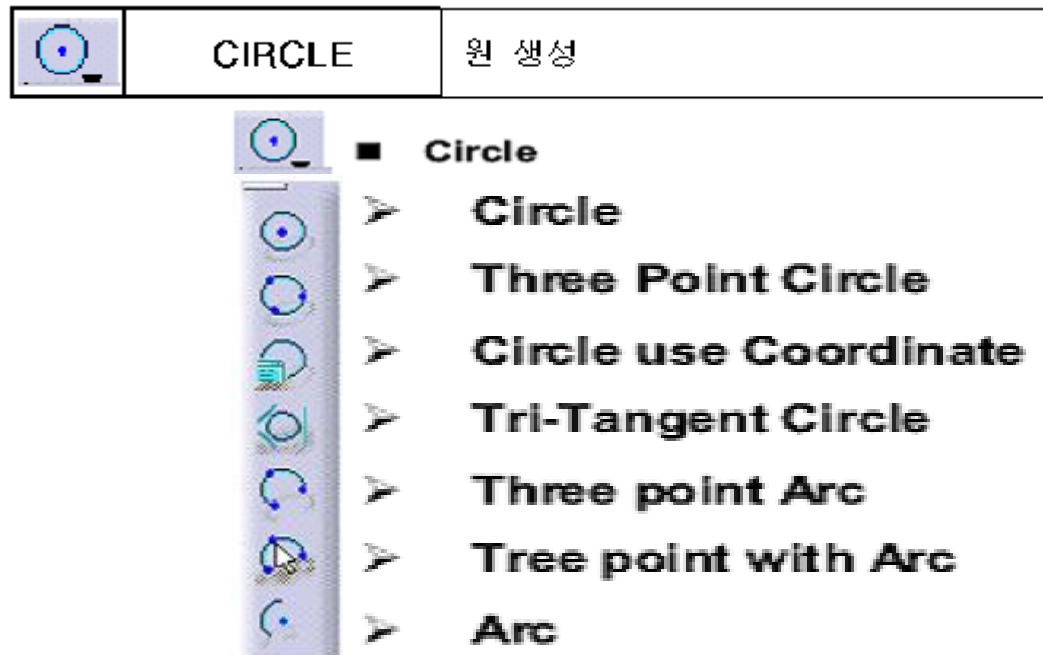
IV. SKETCHER

3-2. Rectangle (2)



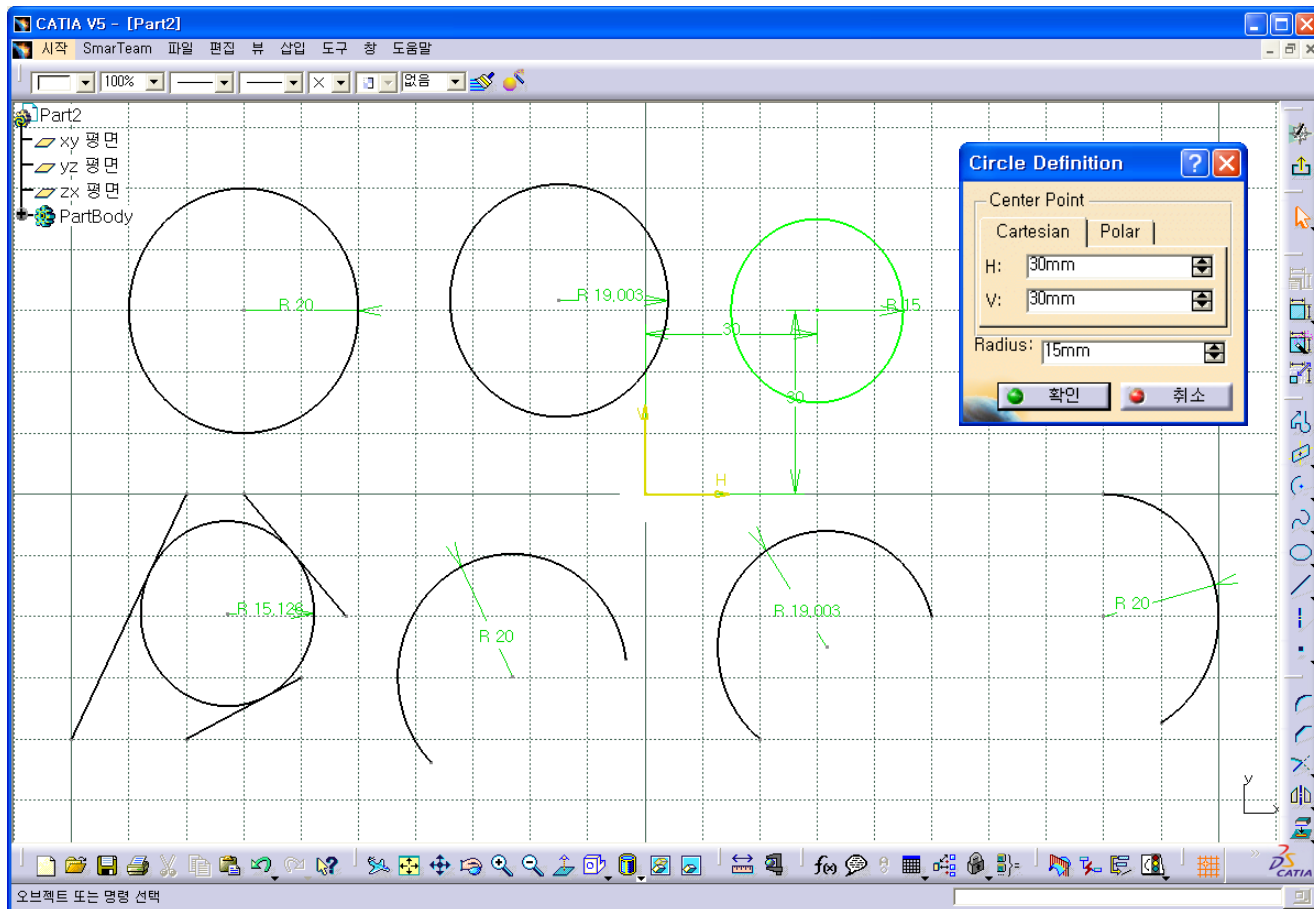
IV. SKETCHER

3-3. Circle (1)



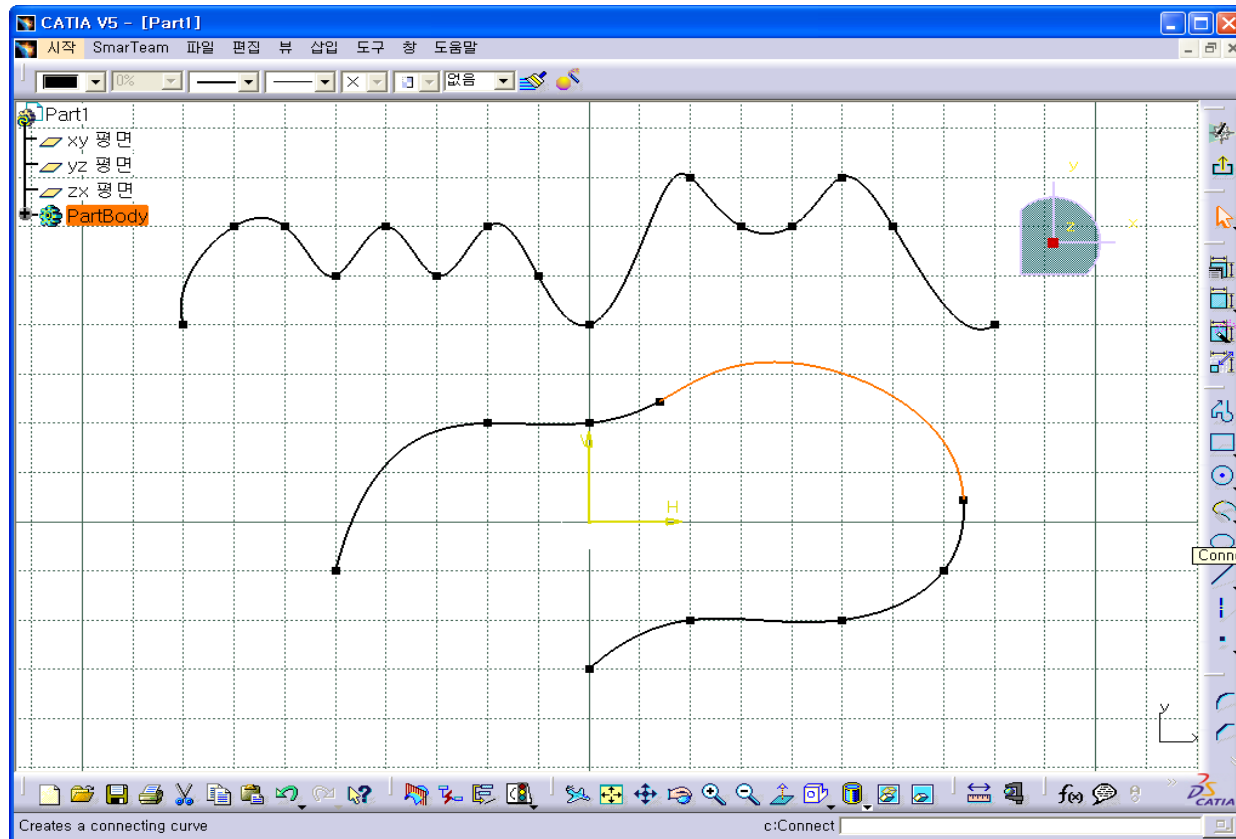
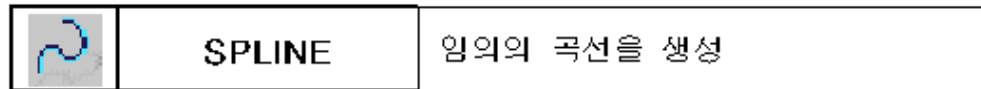
IV. SKETCHER

3-3. Circle (2)



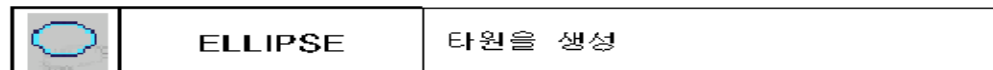
IV. SKETCHER

3-4. Spline



IV. SKETCHER

3-5. Ellipse



I



Ellipse



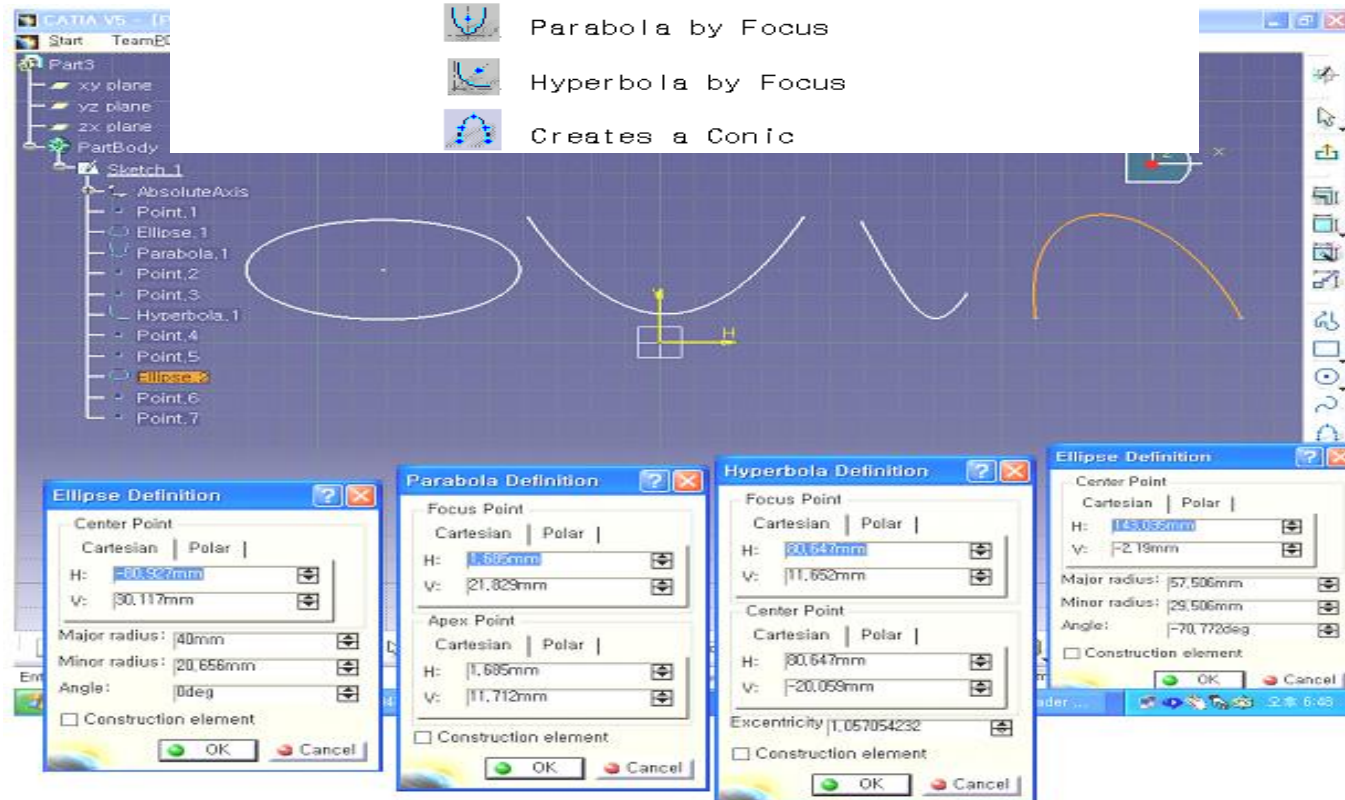
Parabola by Focus



Hyperbola by Focus

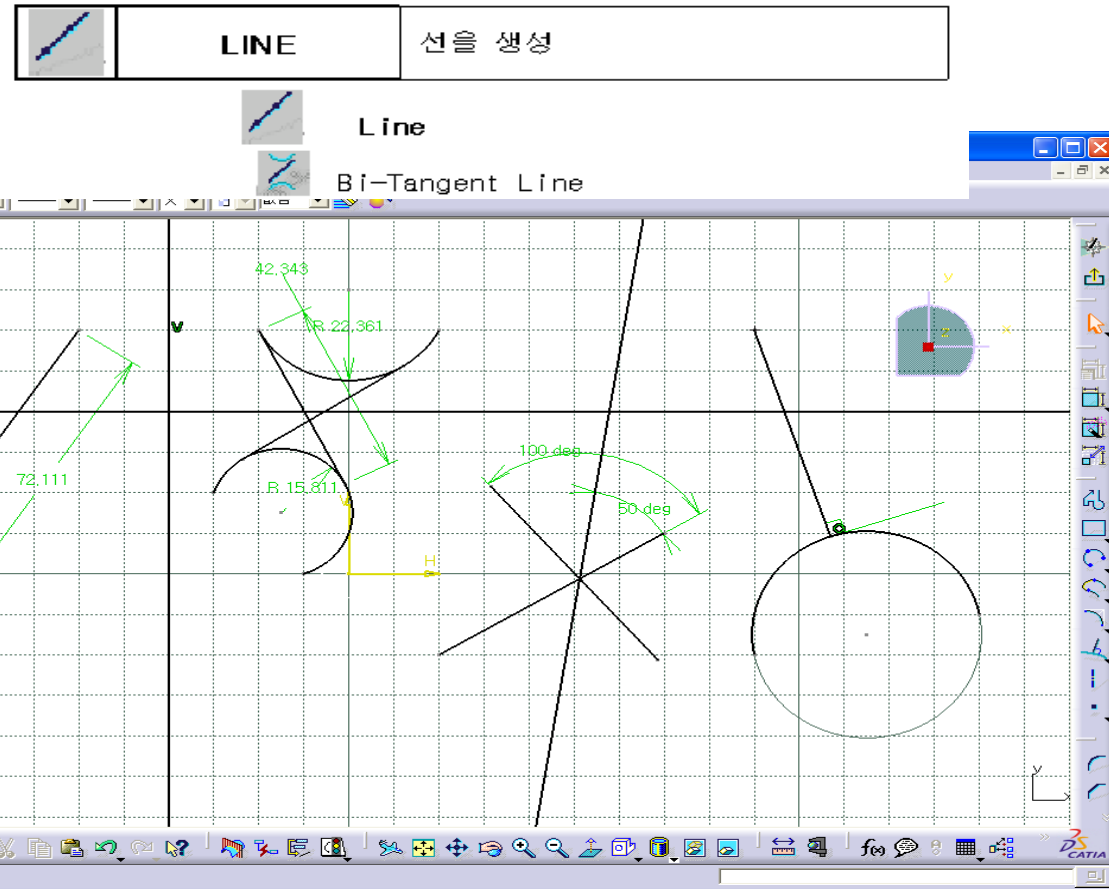


Creates a Conic



IV. SKETCHER

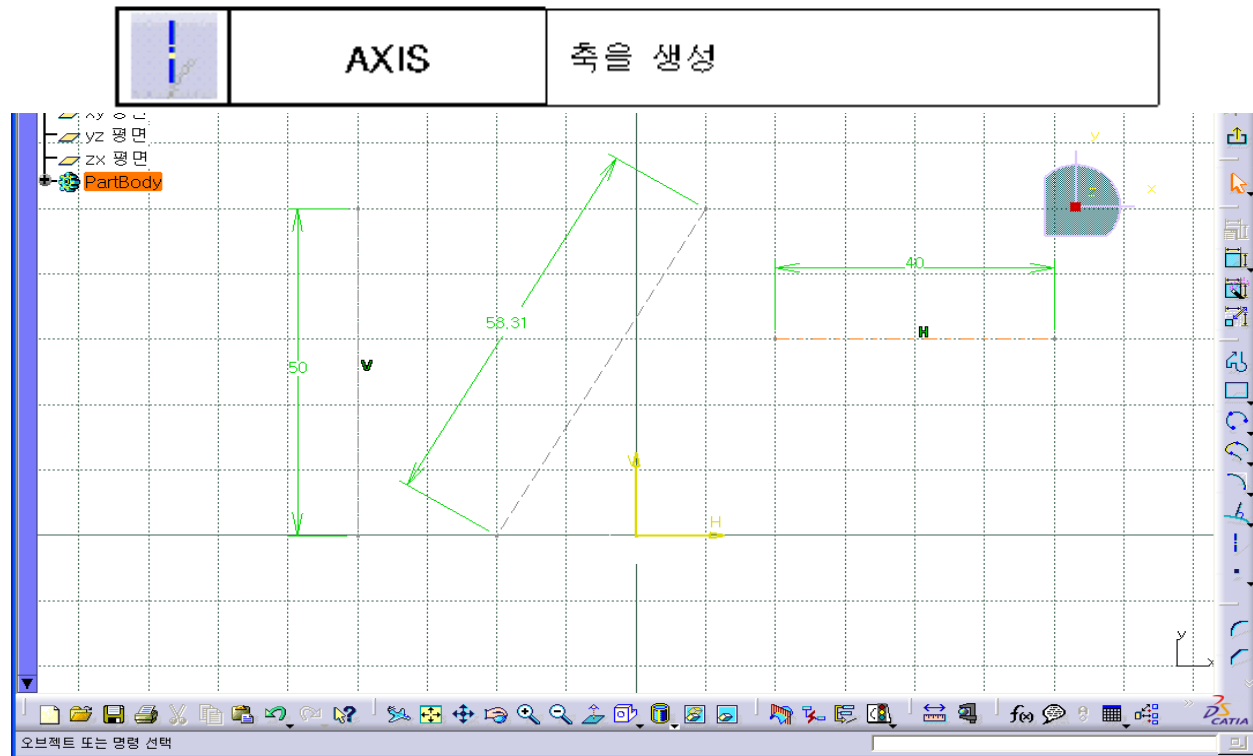
3-6. Line



IV. SKETCHER

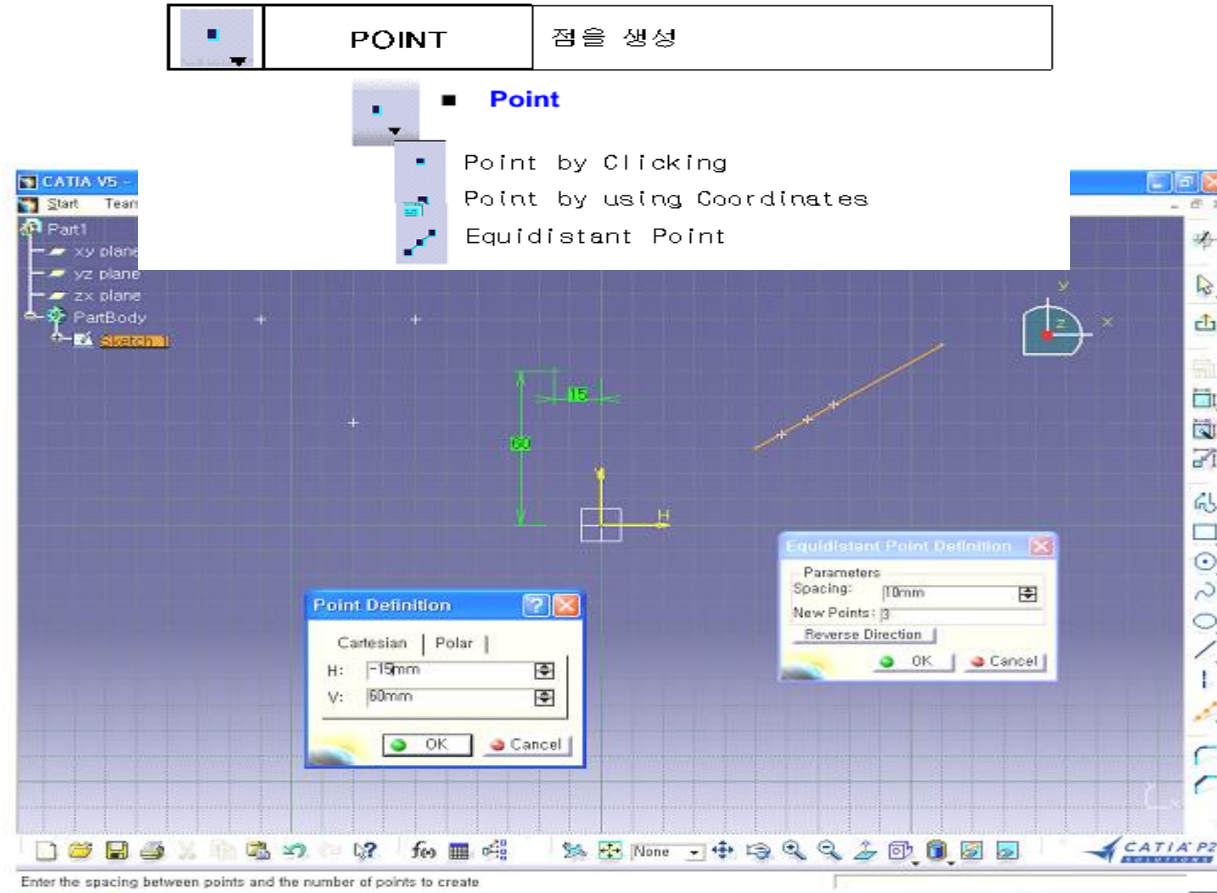
3-7. Axis

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



IV. SKETCHER

3-8. Point



IV. SKETCHER

4. Setting Constraints

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

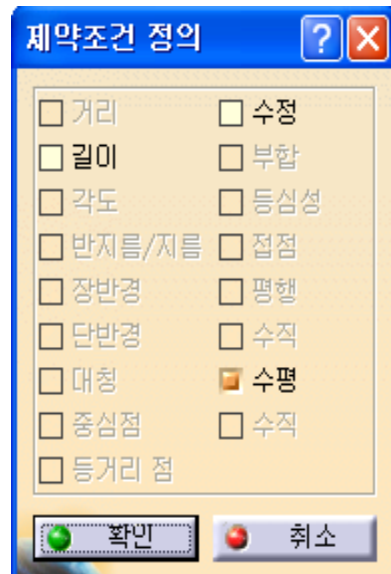
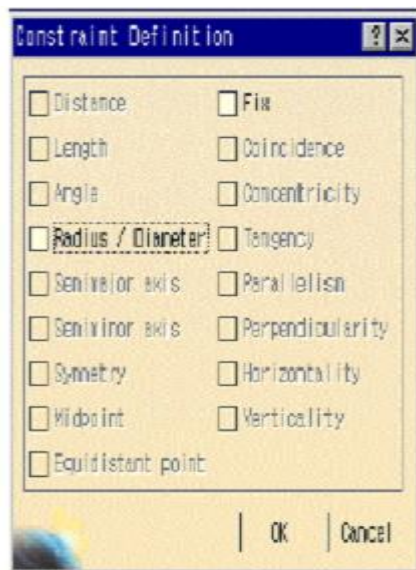


녹 색	정상적인 치수	
짙은 녹색	고정된 치수	치수의 삭제
보라색	필요치 않은 치수 생성	보라색의 치수 중 하나를 삭제
갈 색	부정확한 구속 조건	구속 조건의 삭제

IV. SKETCHER

4-1. Constraints Defined in Dialogue Box

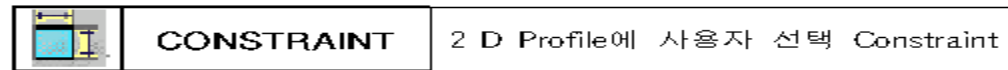
	CONSTRAINTS VIA DIALOGUE BOX	2D Profile에서 Constraint 값을 Dialogue Box에서 정의
--	---	--



- **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 :**



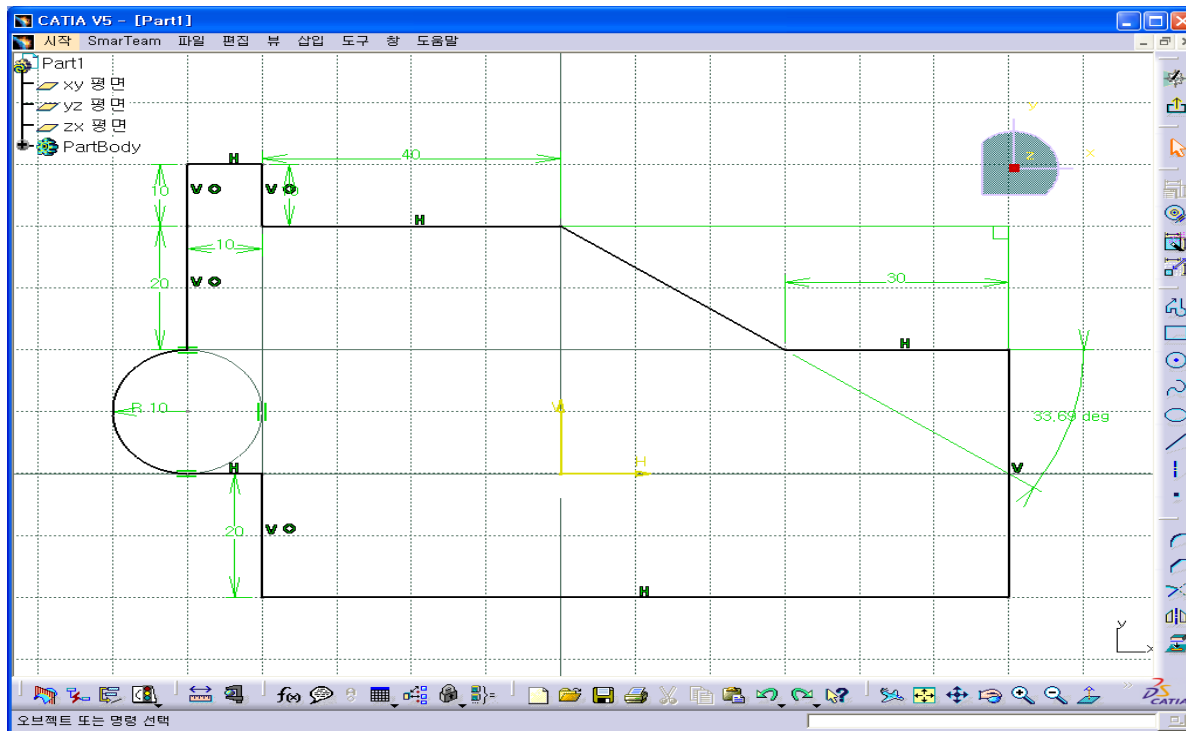
■ **Contact Constraint**



IV. SKETCHER

4-3. Auto Constraint

	AUTO CONSTRAINT	자동으로 Constraint 부여
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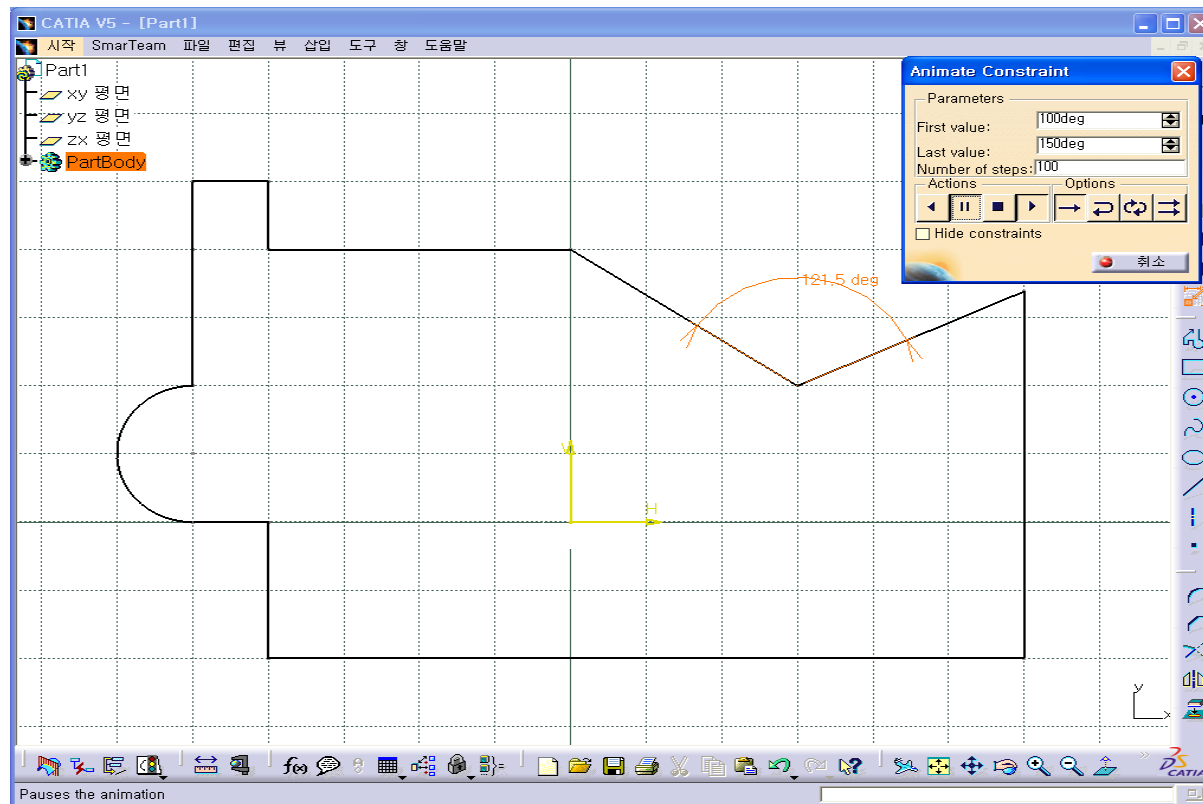
IV. SKETCHER

4-4. Animate Constraint



**ANIMATE
CONSTRAINT**

Constraint에 부여된 Value를 자동으로 조절하면서 최상의 Value를 생성



IV. SKETCHER

5. Operations on Profiles

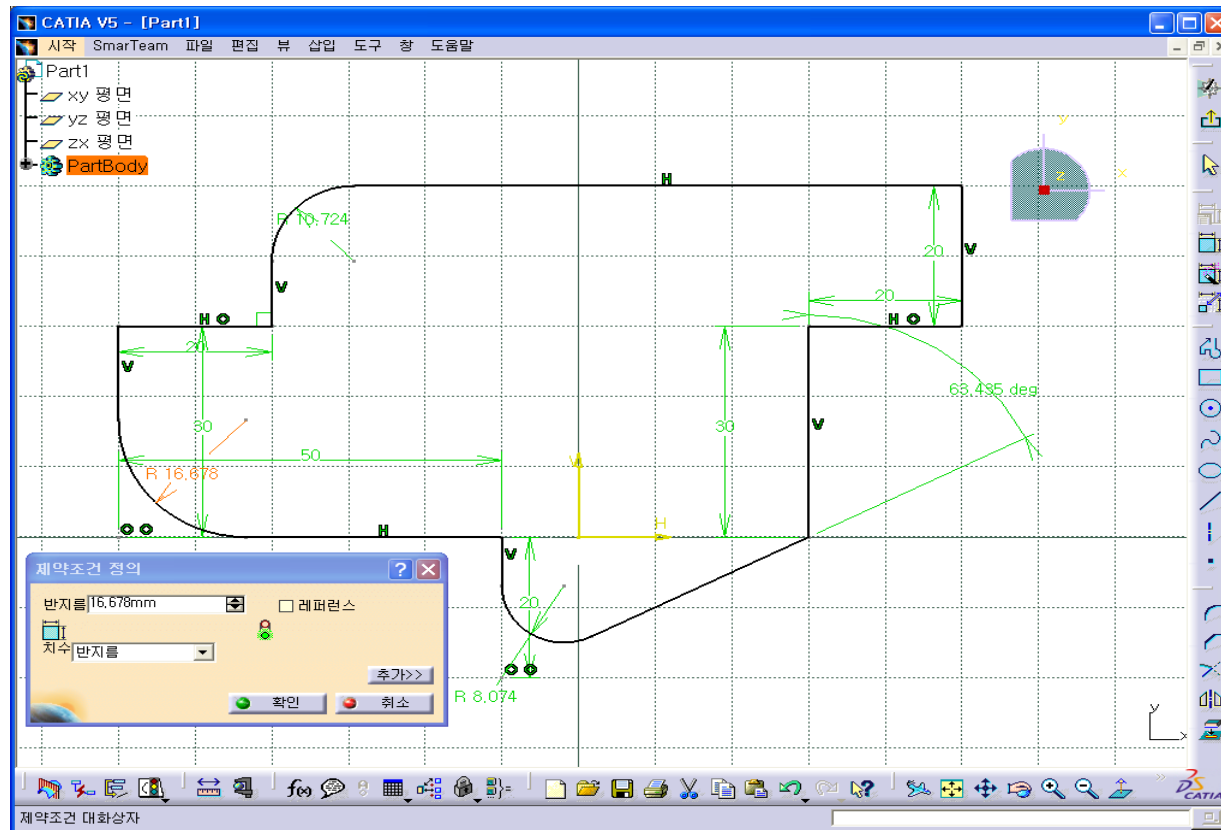
생성한 Profile 의 편집이나 이동, 복사 등의 기능을 모아
놓은 툴 바이다.



IV. SKETCHER

5-1. Corner

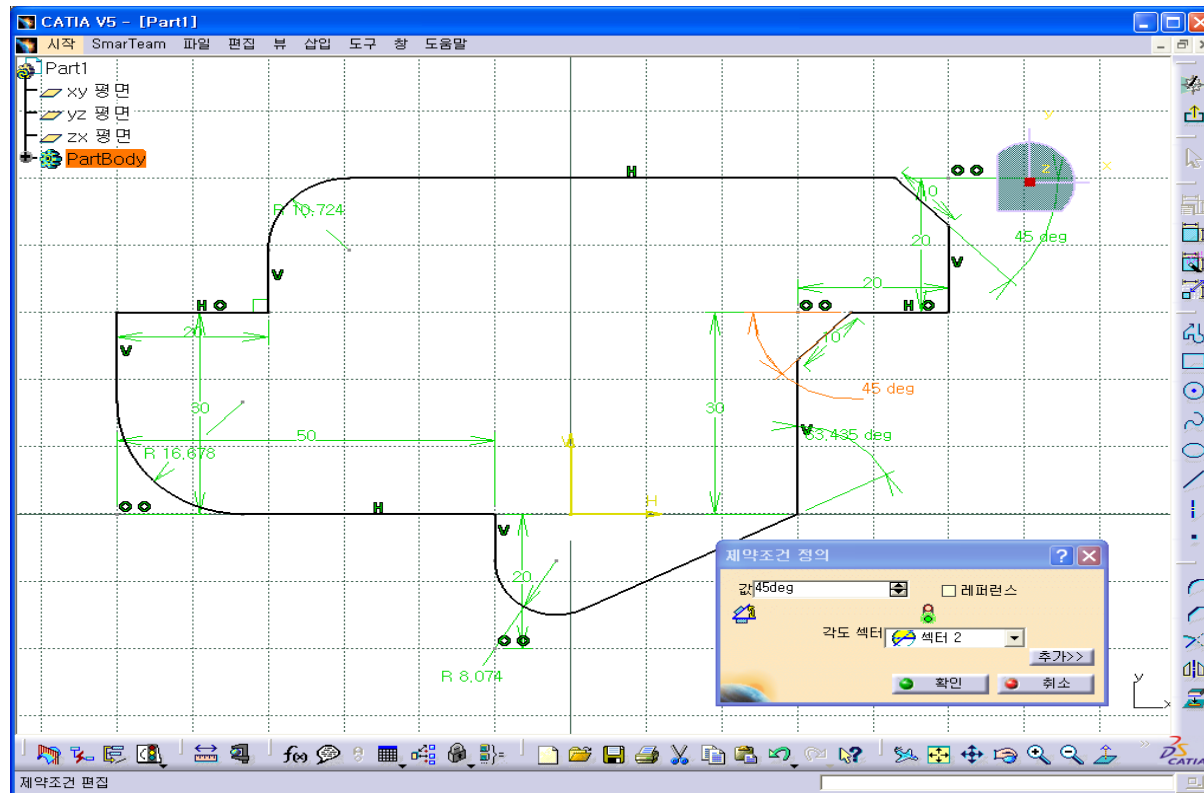
	CORNER	Corner 처리시 사용하는 기능
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IV. SKETCHER

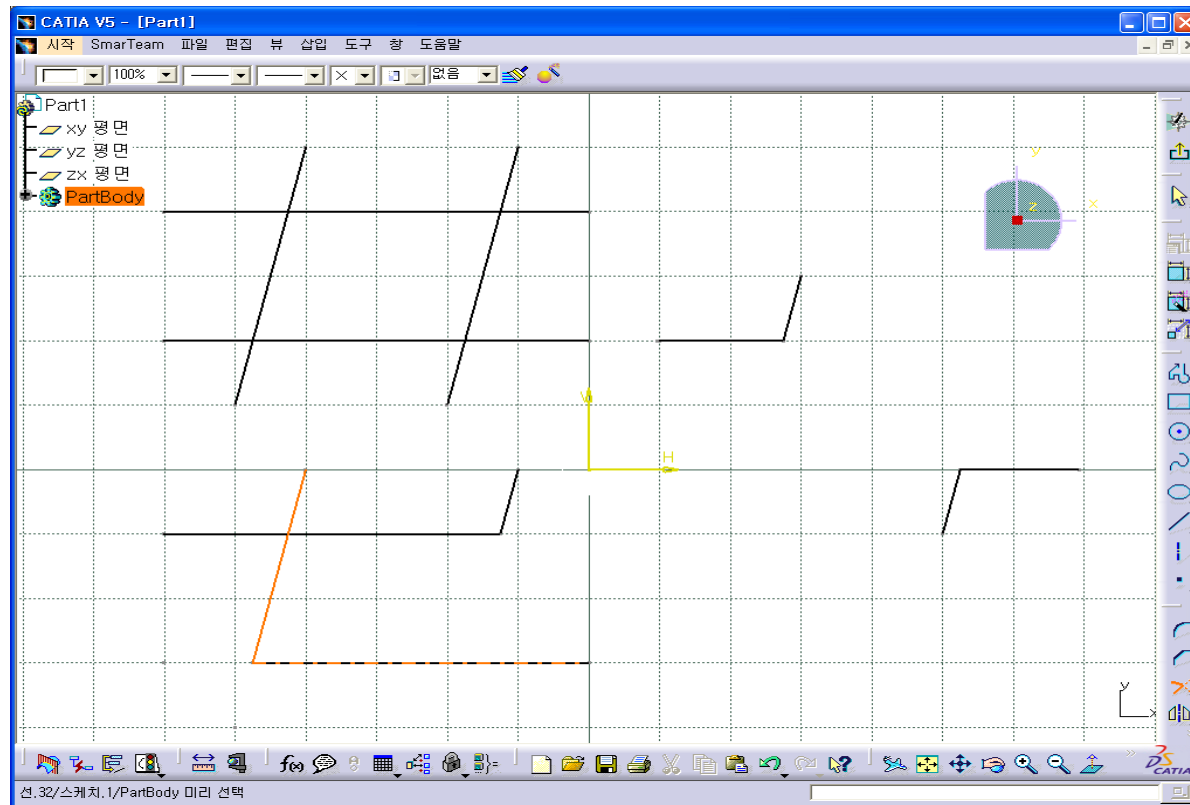
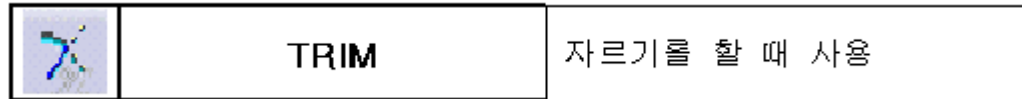
5-2. Chamber

	CHAMFER	모따기 작업시 사용하는 기능
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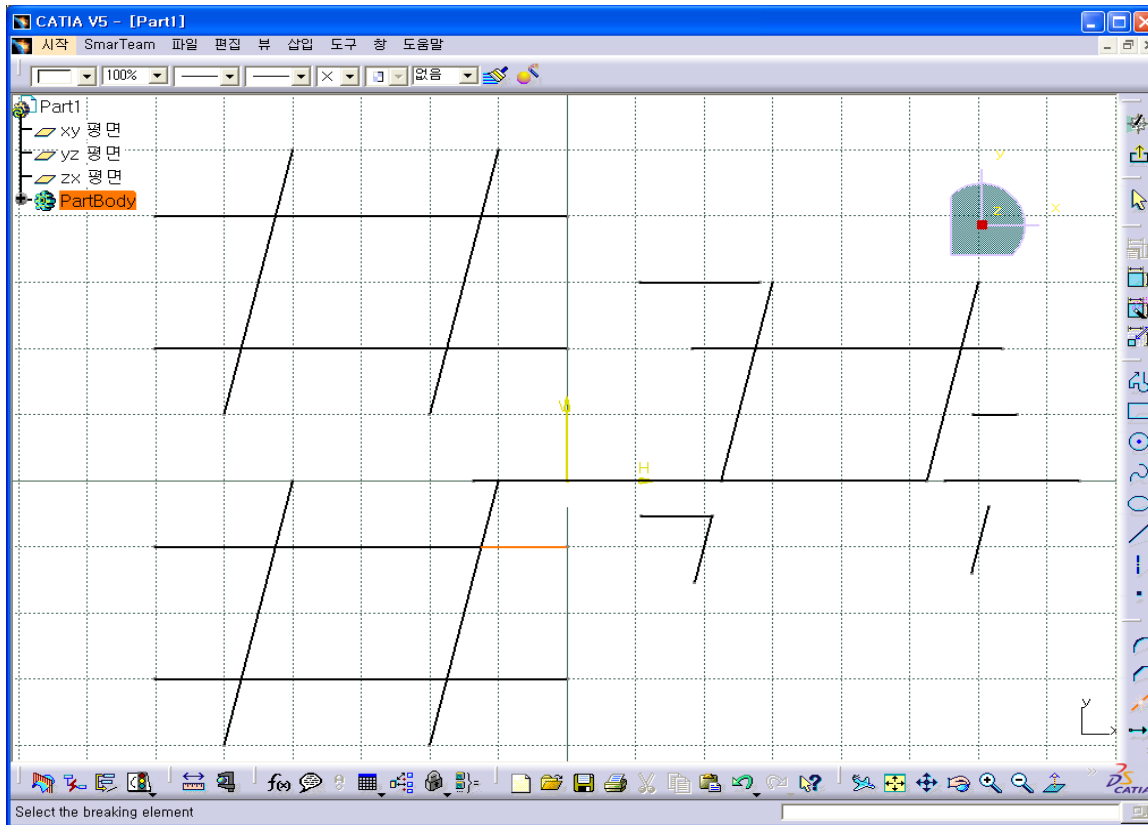
IV. SKETCHER

5-3. Trim, Quick Trim



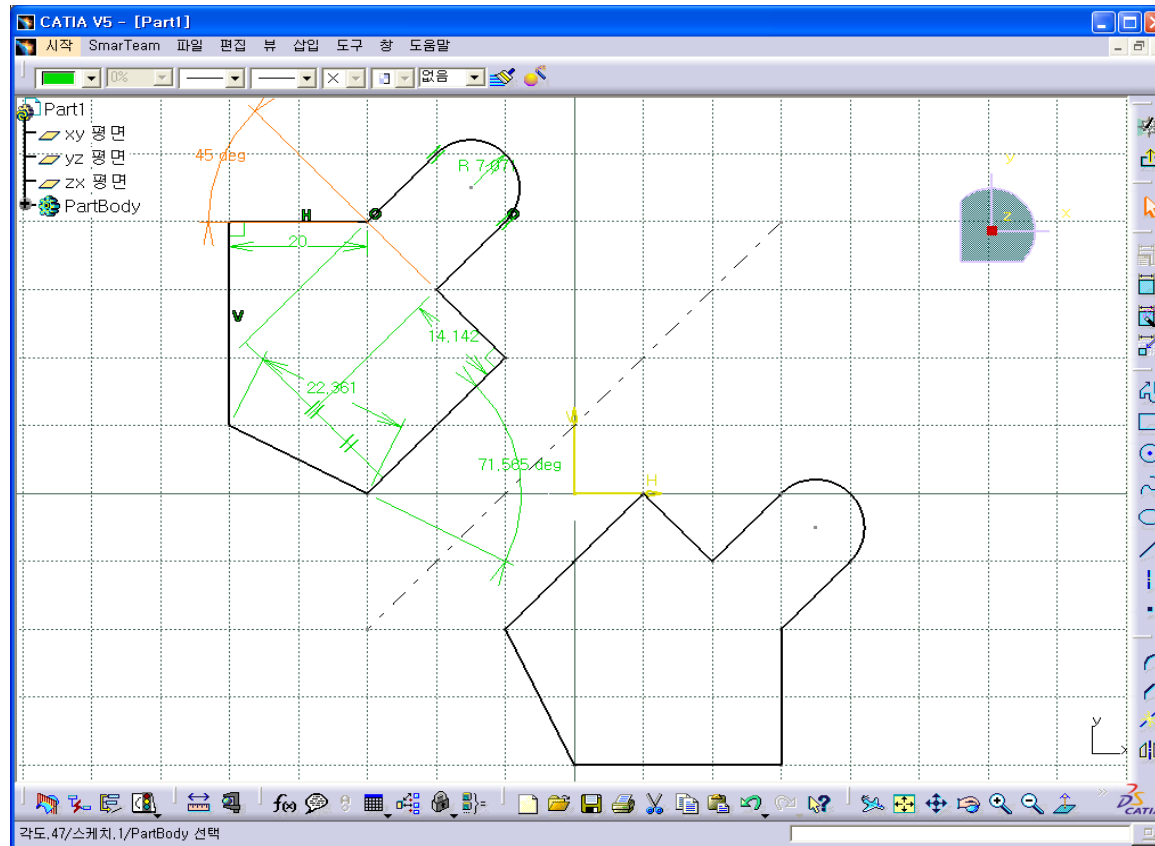
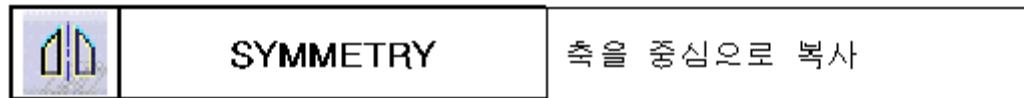
IV. SKETCHER

5-4. Break, Close, Complement



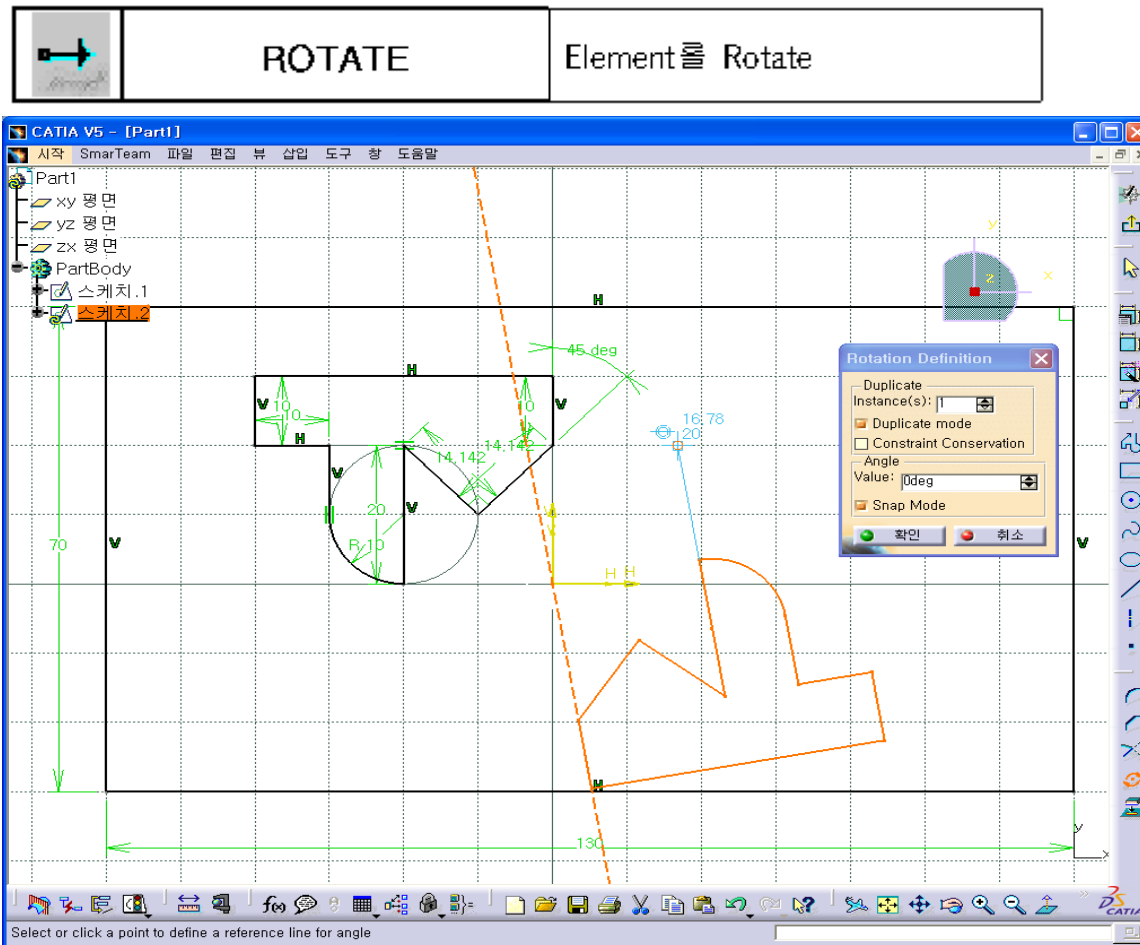
IV. SKETCHER

5-5. Mirror, Symmetry



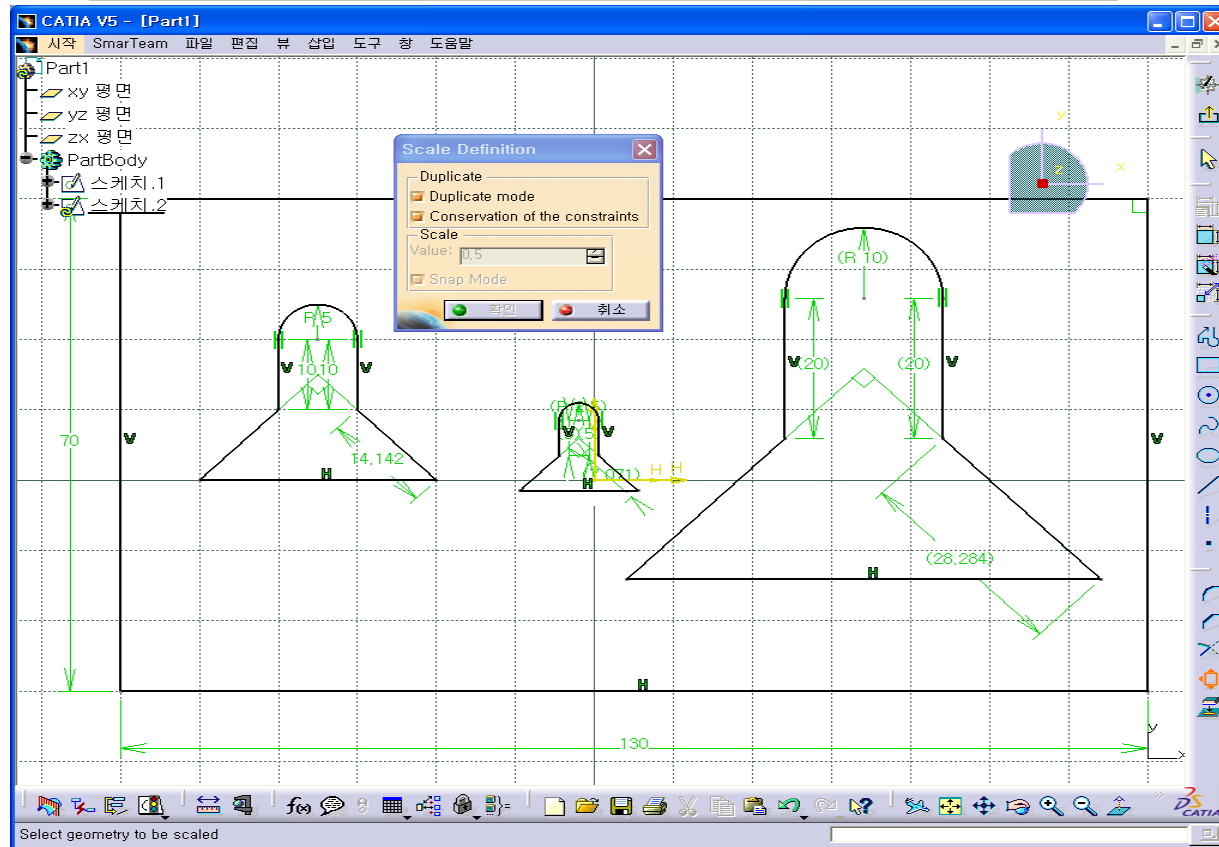
IV. SKETCHER

5-7. Rotate



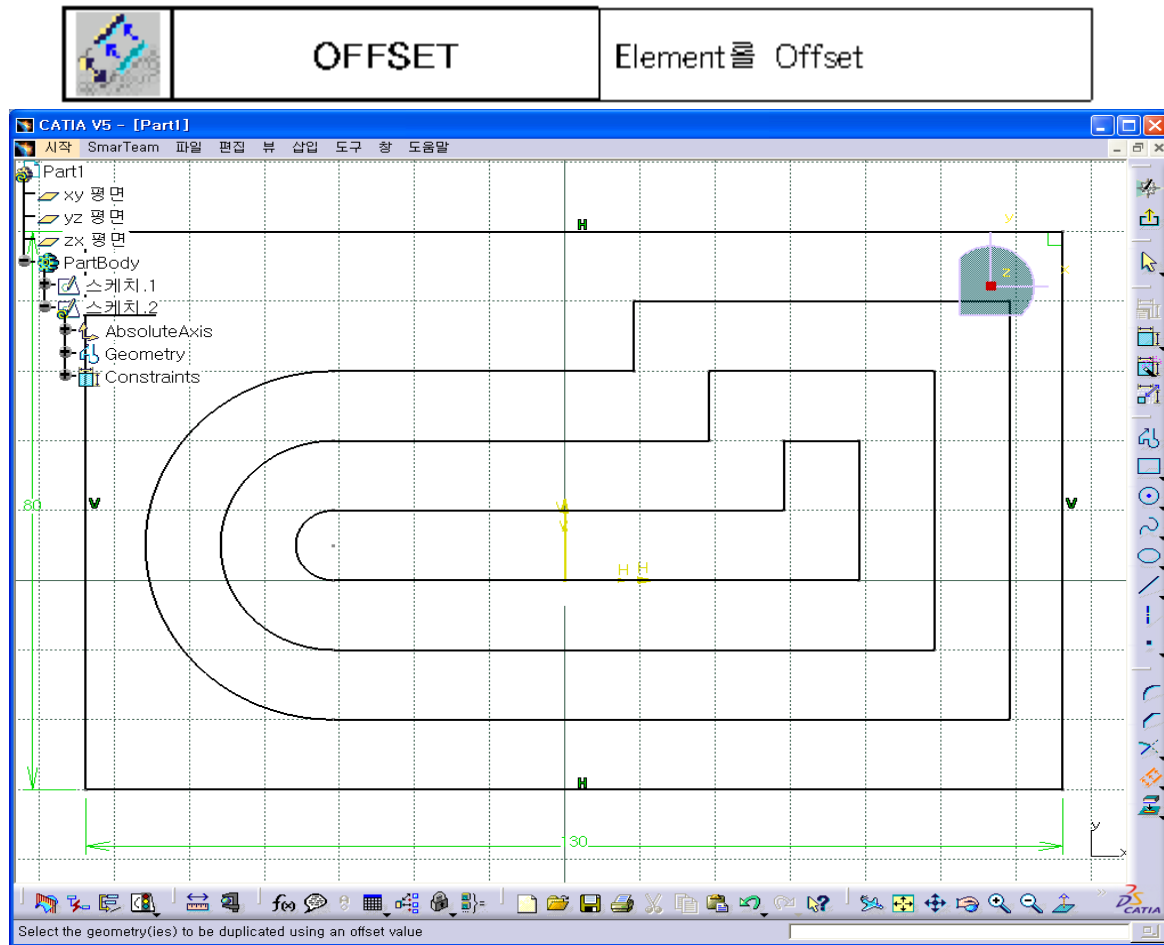
IV. SKETCHER

5-8. Scale



IV. SKETCHER

5-9. Offset




IV. SKETCHER


5-10. Project 3D Elements

	PROJECT 3D ELEMENT	Plane 상에 3D Element를 투영
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5-11. Intersect 3D Elements

	INTERSECT 3D ELEMENT	Plane 상에서 3D Element들의 Intersect element <u>를</u> 생성
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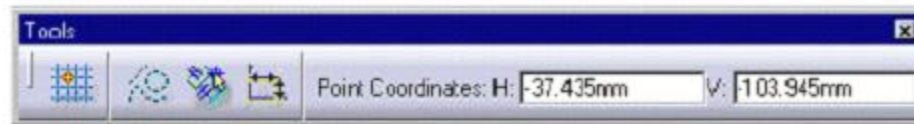
5-12. Isolate





	ISOLATE	isolating
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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

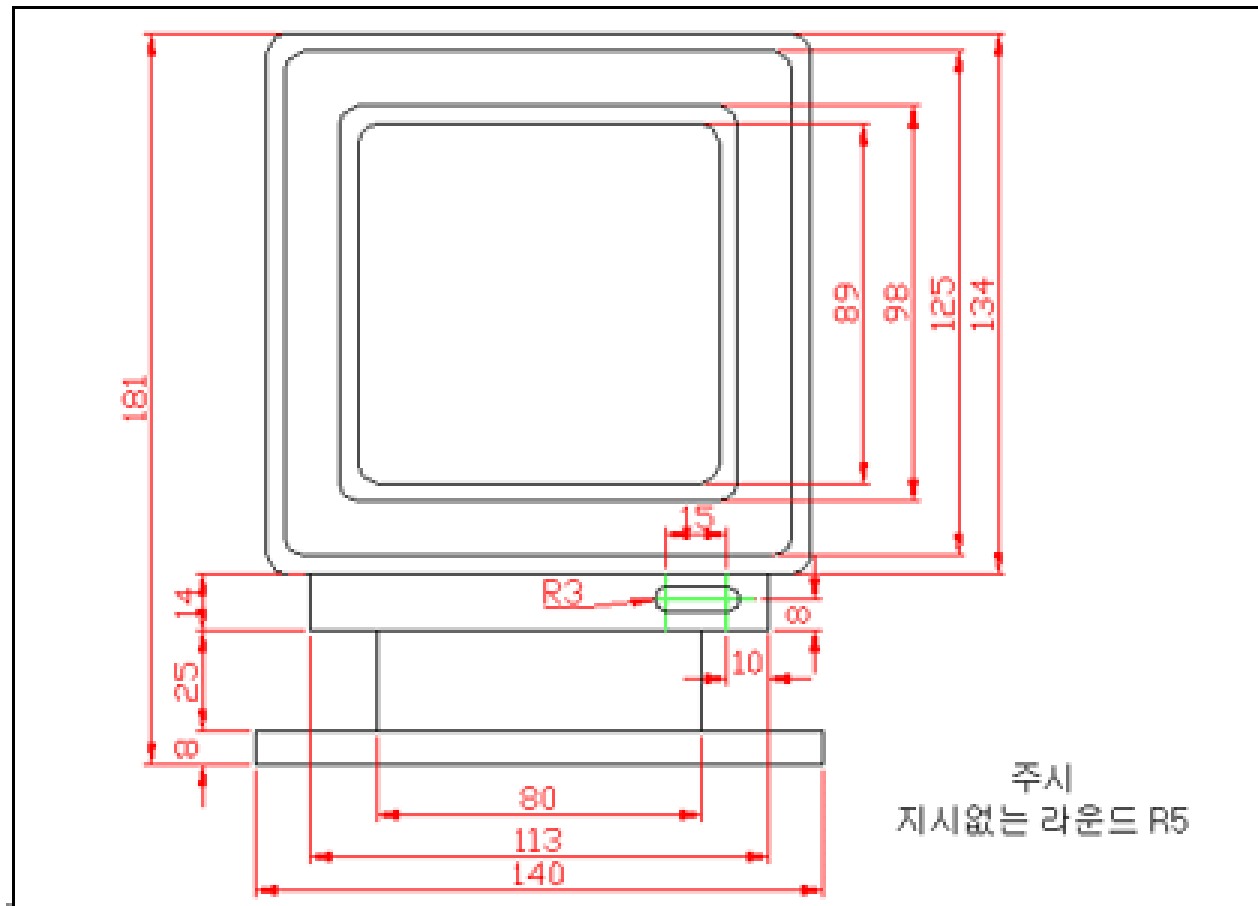
치수와 관계된 구속조건들을 생성 시켜준다.

6-4. Dimensional Constraints

치수와 관계된 구속조건들을 생성 시켜준다.

IV. SKETCHER

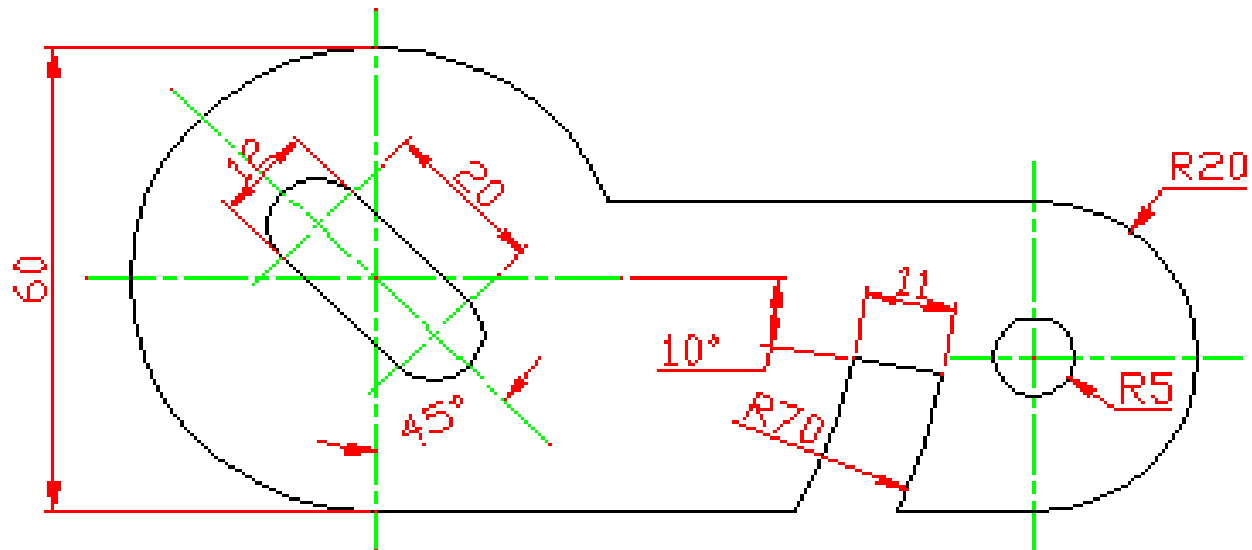
7. 예 제도면 (1)



IV. SKETCHER

7. 예 제도면 (2)

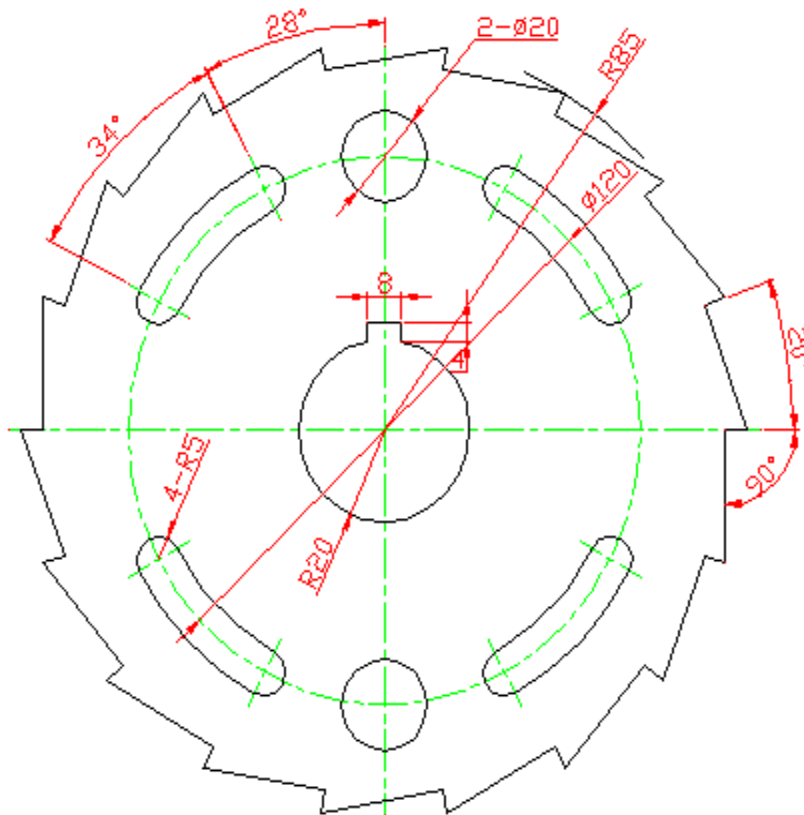
ARC, 상대극 좌표



IV. SKETCHER

7. 예 제도면 (3)

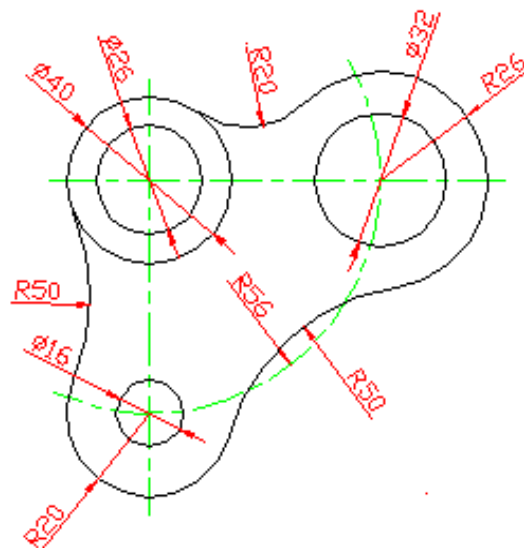
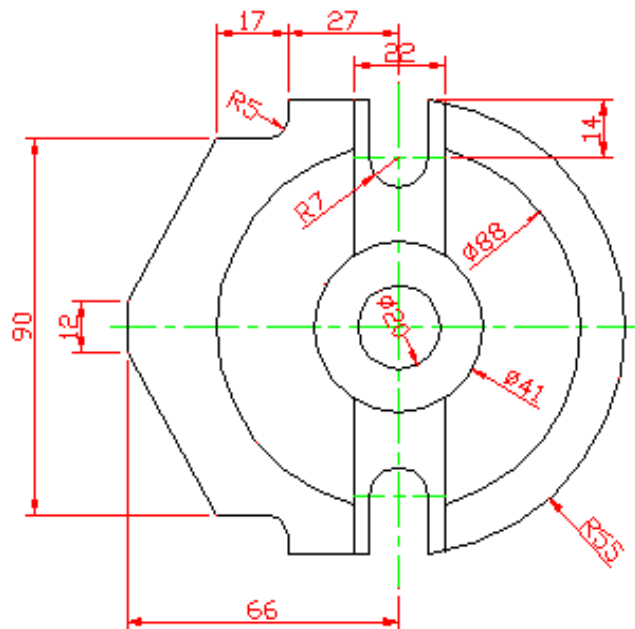
Array, 상대극좌표, Mirror



IV. SKETCHER

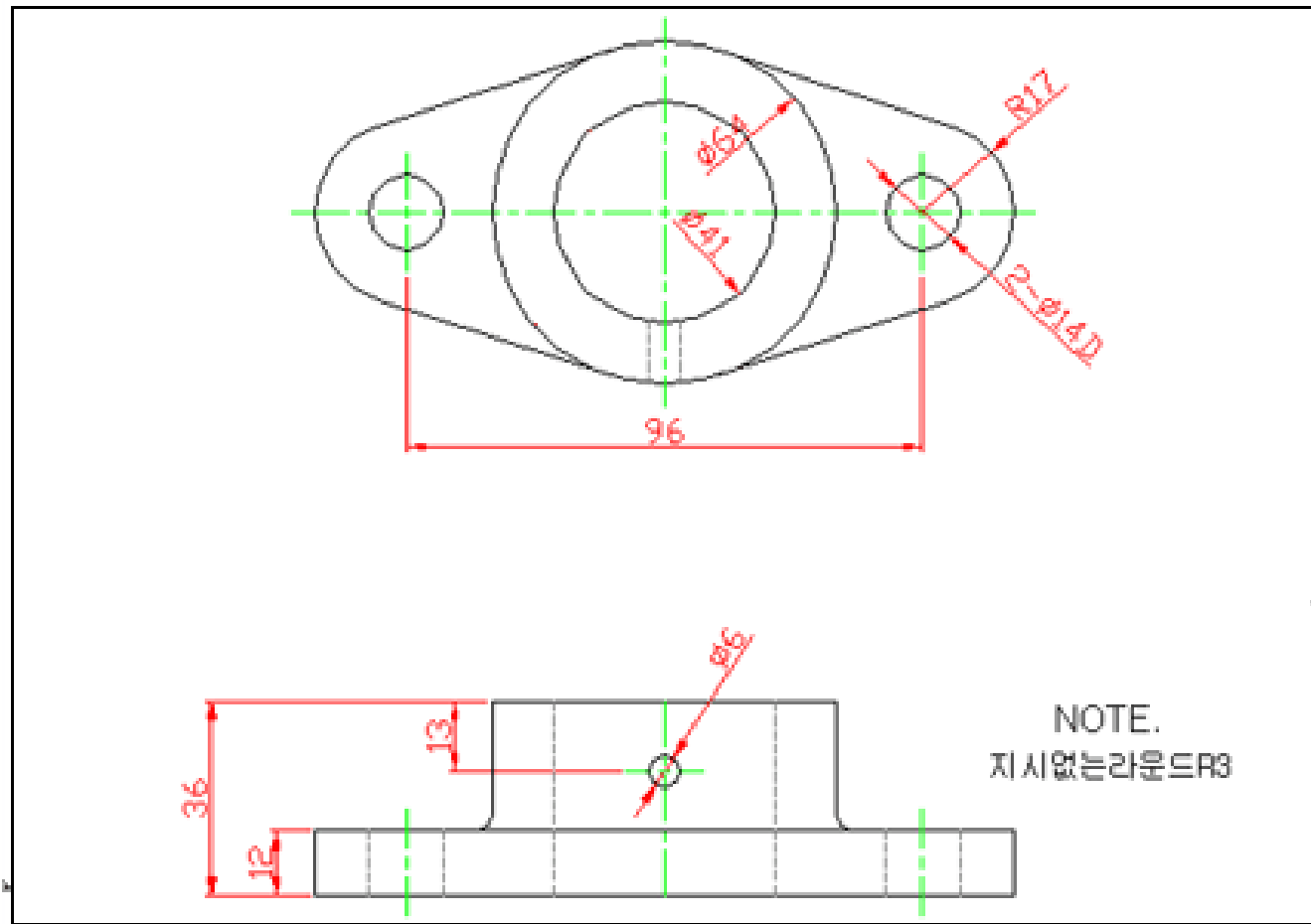
7. 예 제도면 (4)

MIRROR 및 CIRCLE, FILLET 응용



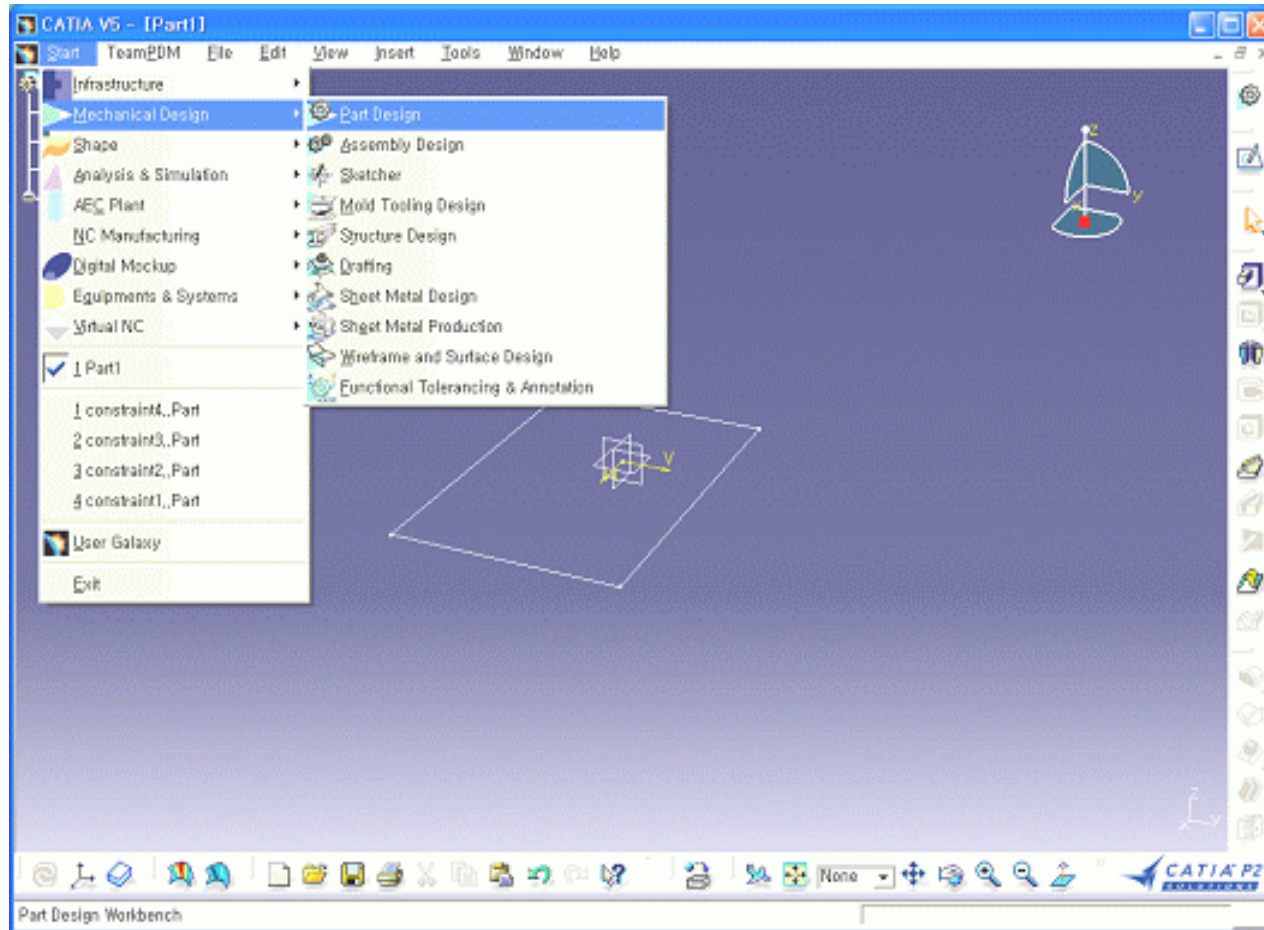
IV. SKETCHER

7. 예 제도면 (5)



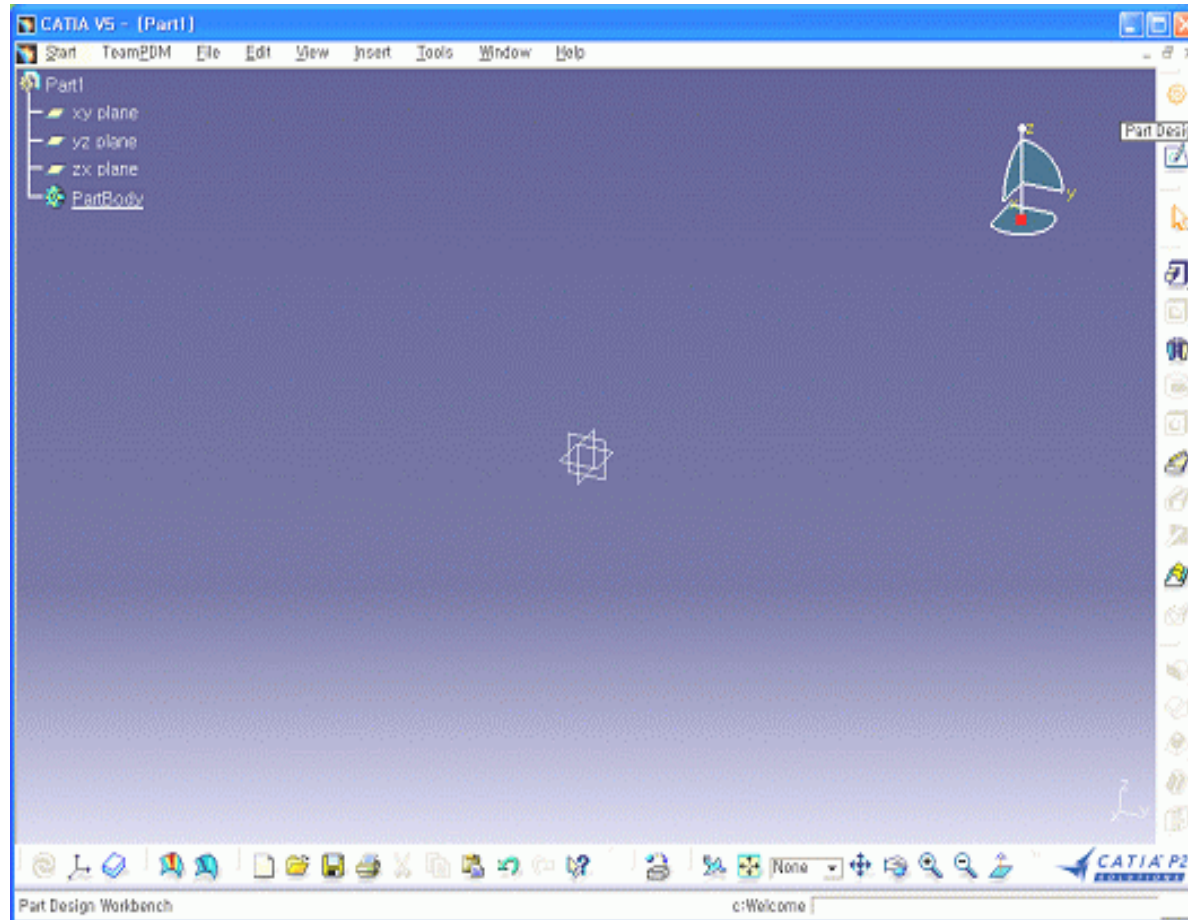
V. PART DESIGN

1. Introduction (1)



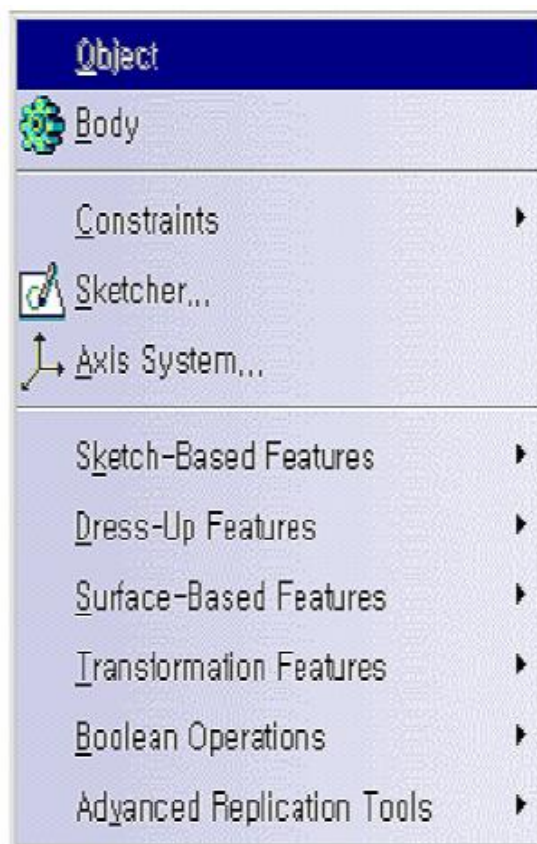
V. PART DESIGN

1. Introduction (2)



V. PART DESIGN

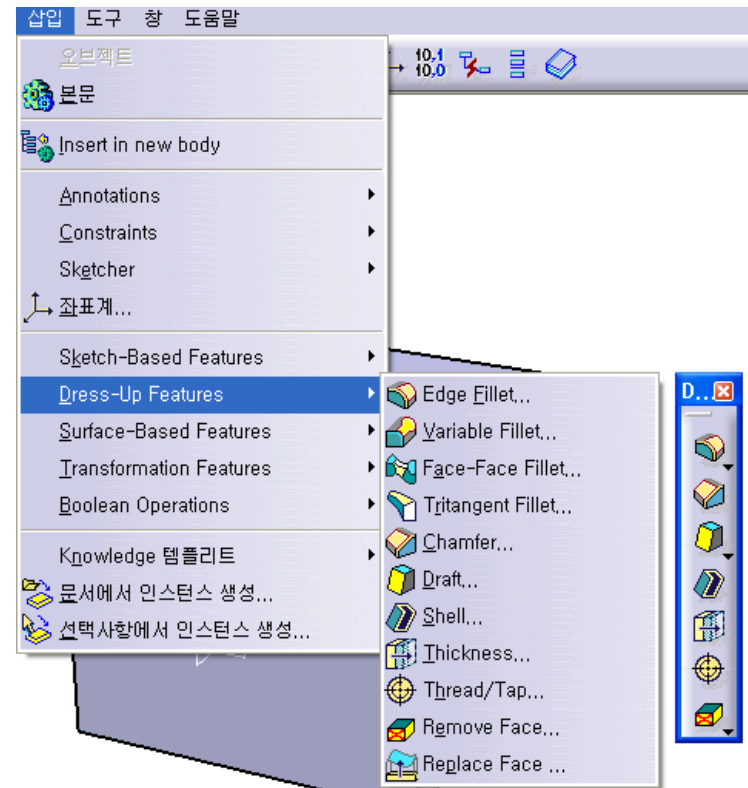
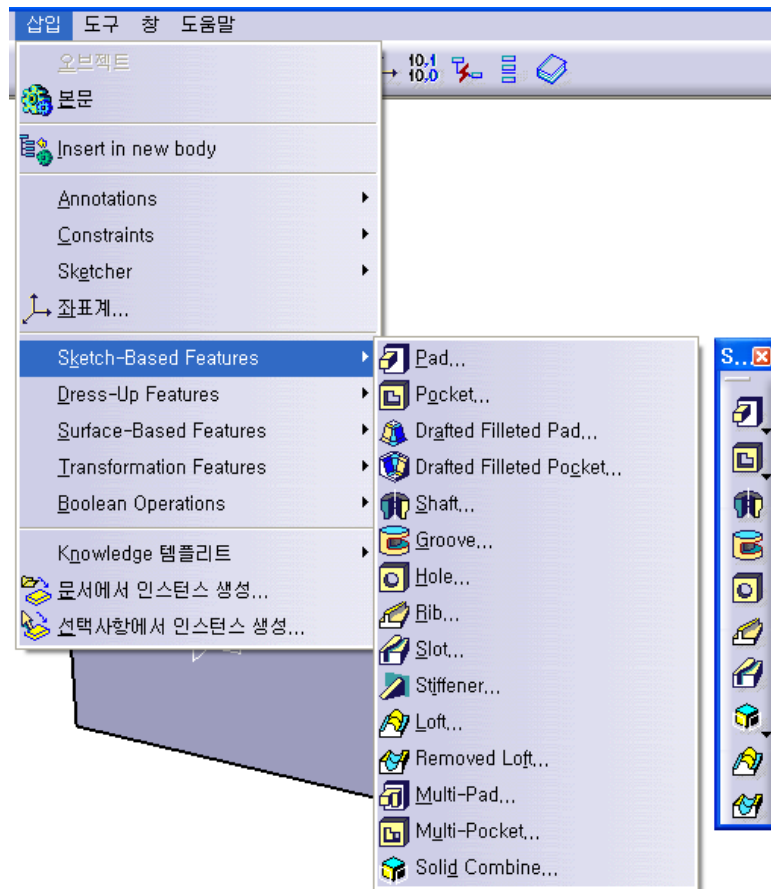
2. Menu bar 와 Icon (1)



- **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** :

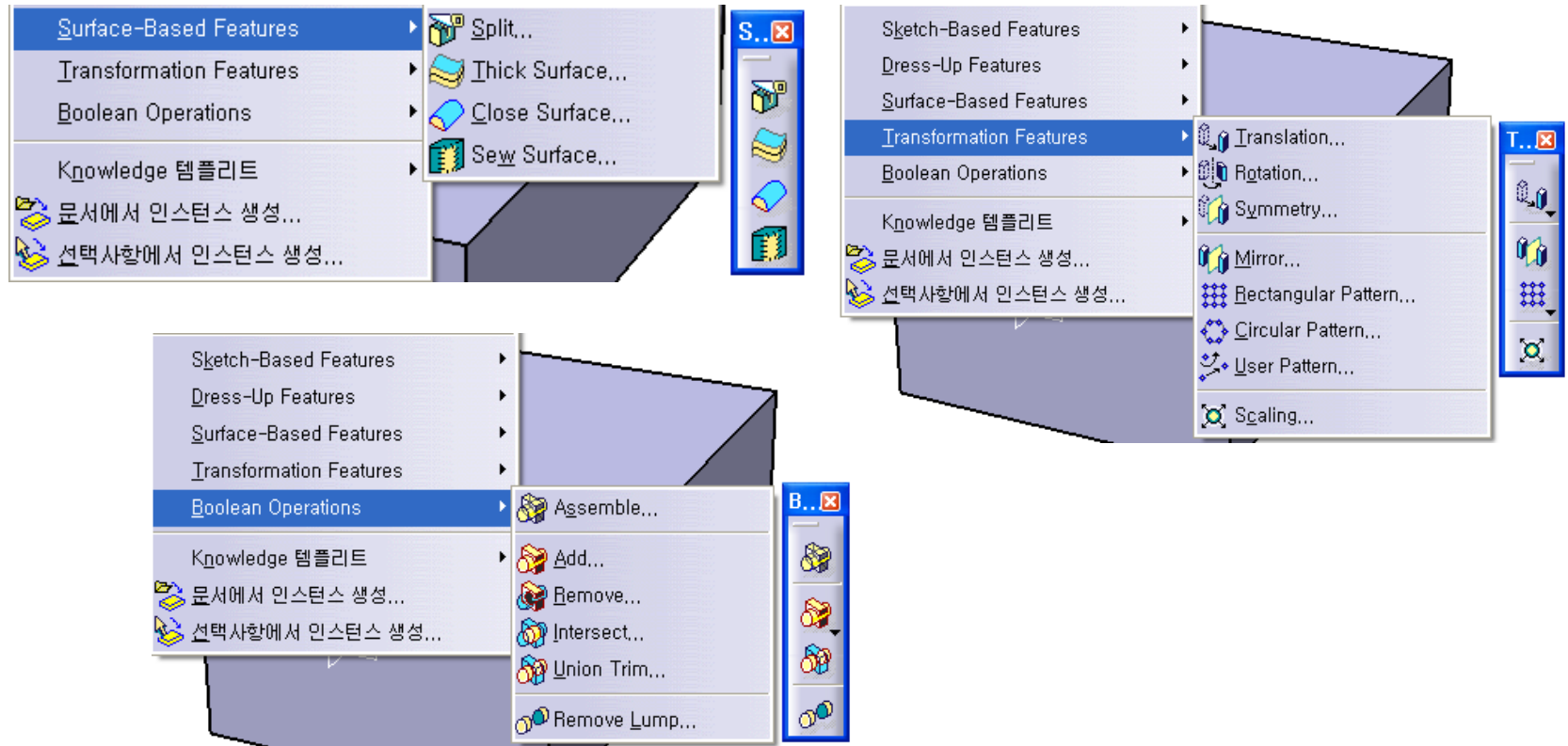
V. PART DESIGN

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



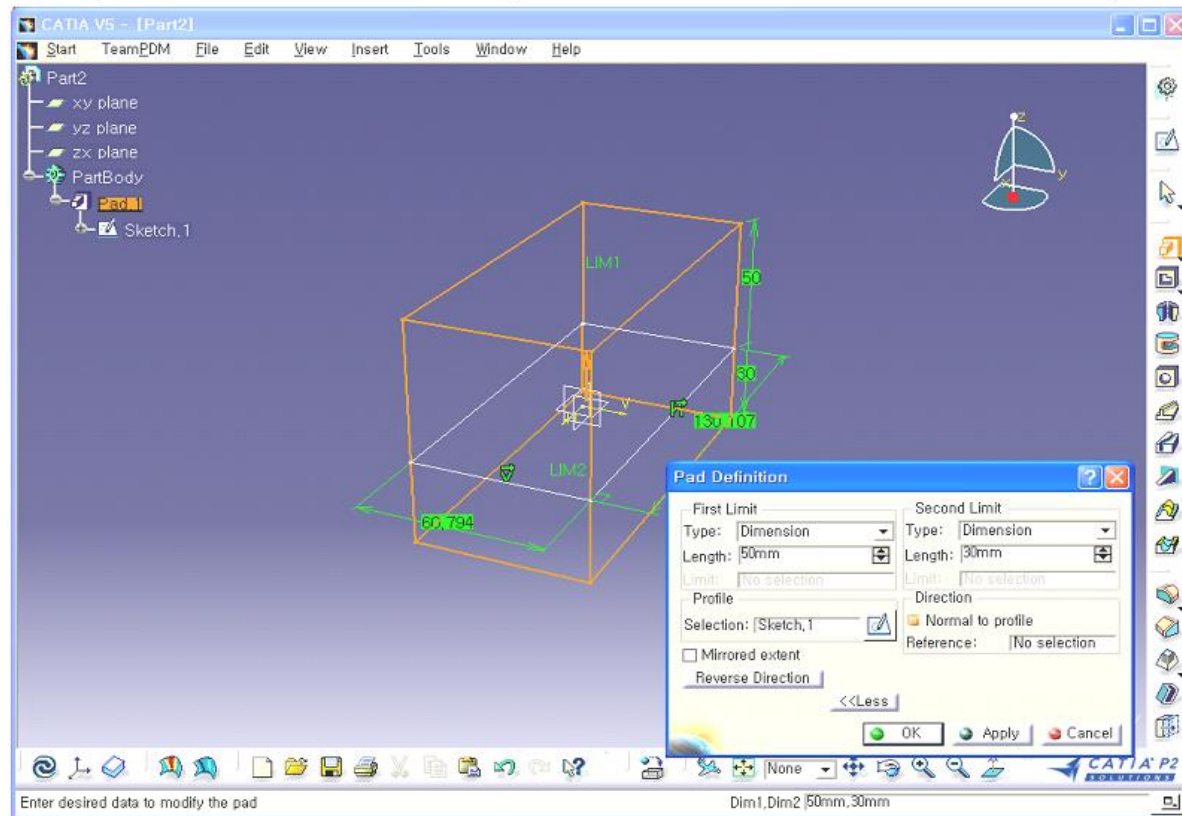
V. PART DESIGN

3-1. Pad



PAD

2 D Profile을 솔리드로 작성



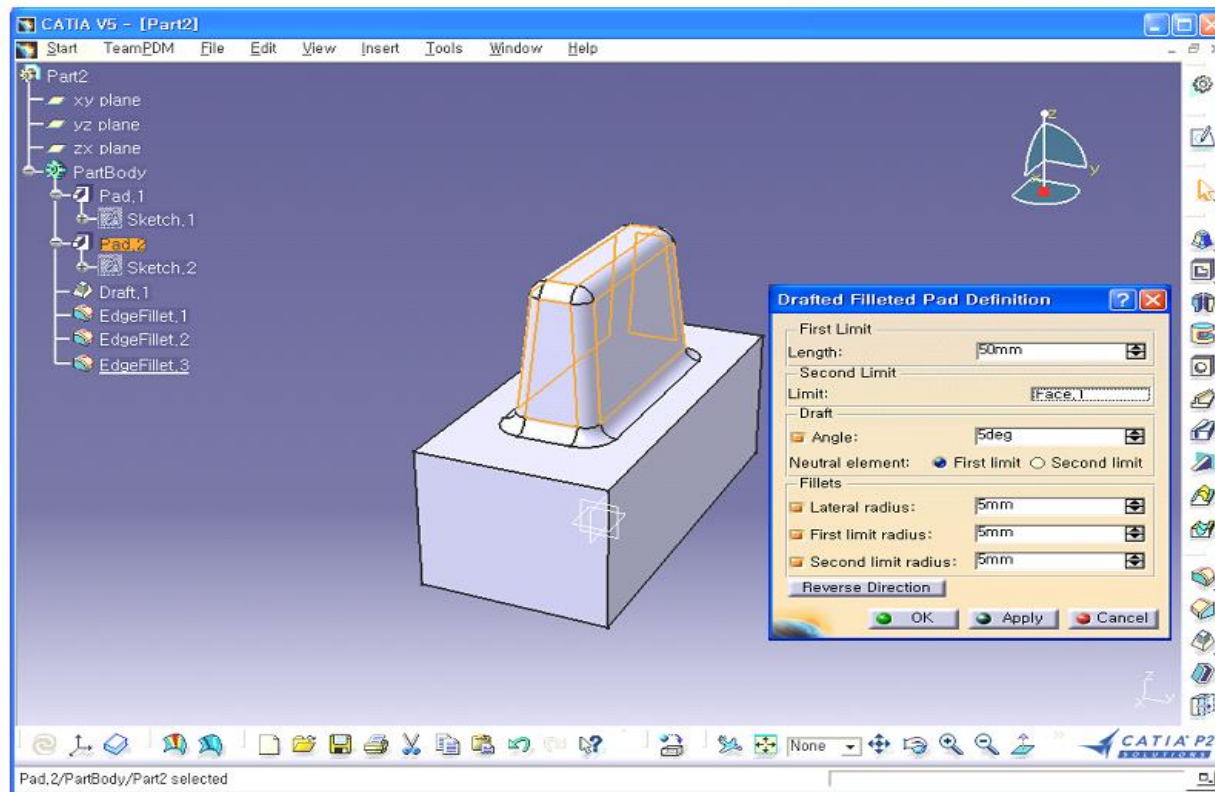
V. PART DESIGN

3-2. Drafted Filleted Pad



**DRAFTED
FILLETED PAD**

Solid에 Drafting 과 Filleting 작업을
동시에 수행 Pad 형 Feature 생성



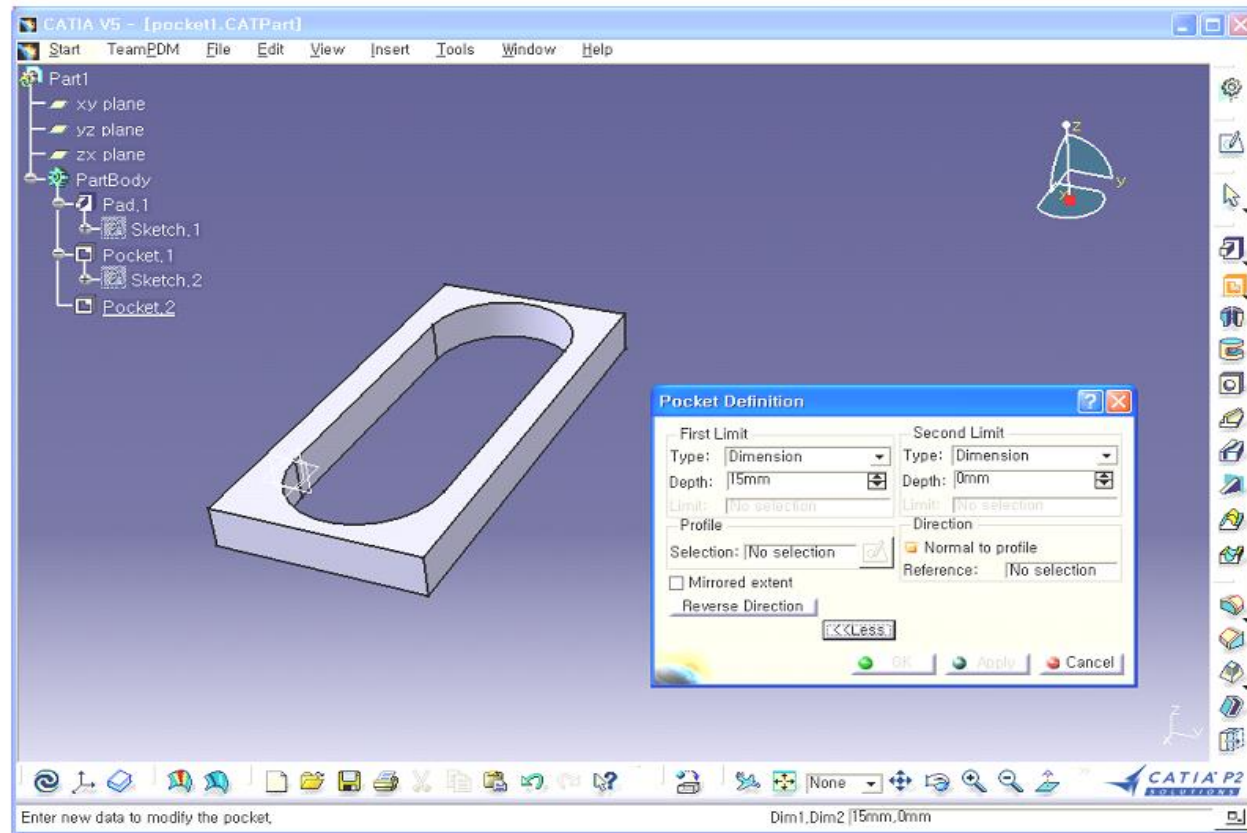
V. PART DESIGN

3-3. Pocket



POCKET

솔리드에 Pocket 생성



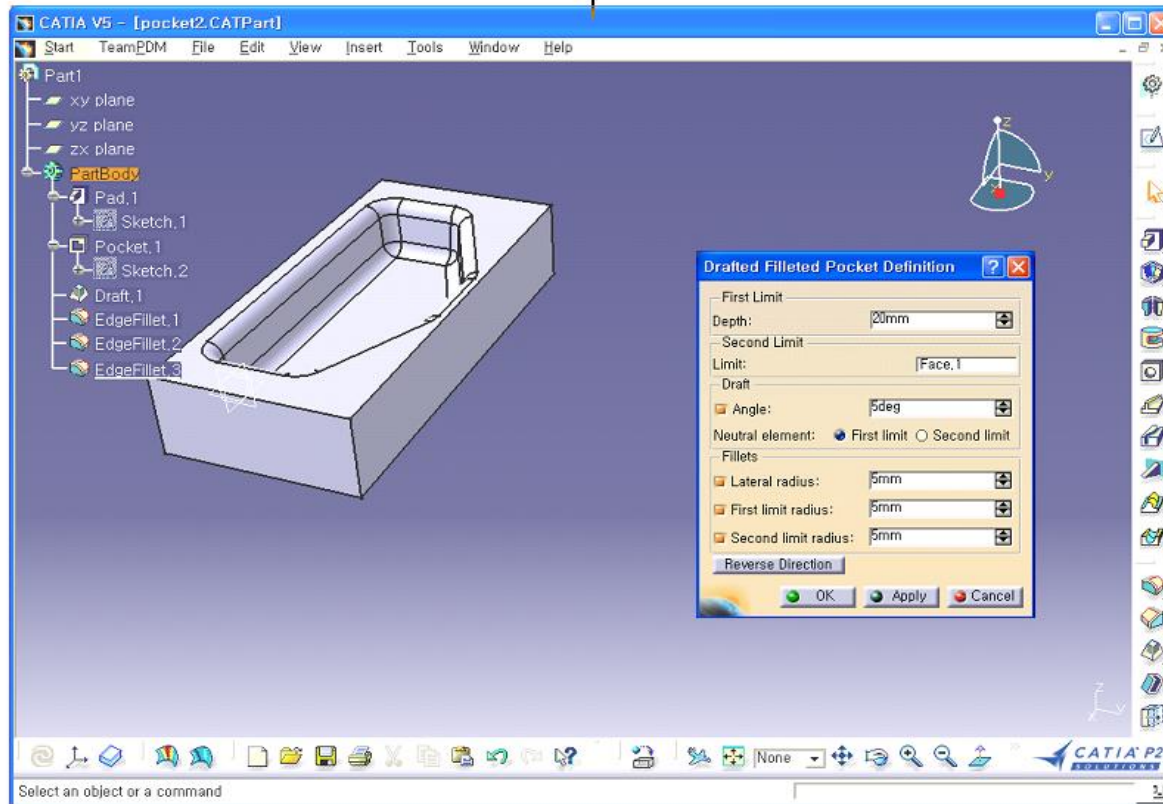
V. PART DESIGN

3-4. Drafted Filleted Pocket




DRAFTED FILLED-ETED POCKET

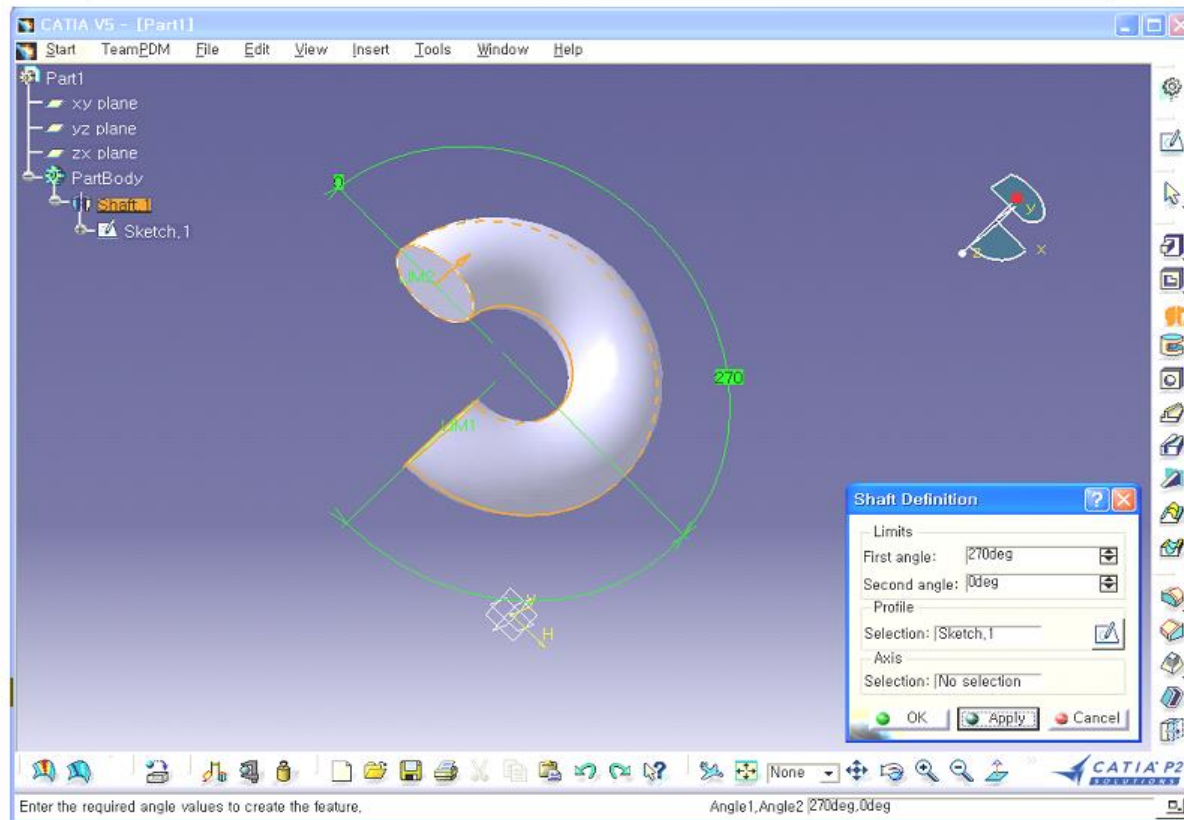
솔리드에 Drafting 과 Filleting 작업을 동시에 수행하면서 Pocket 형 Feature 생성



V. PART DESIGN

3-5. Shaft

	SHAFT	회전형 Solid 형상을 생성하는 기능
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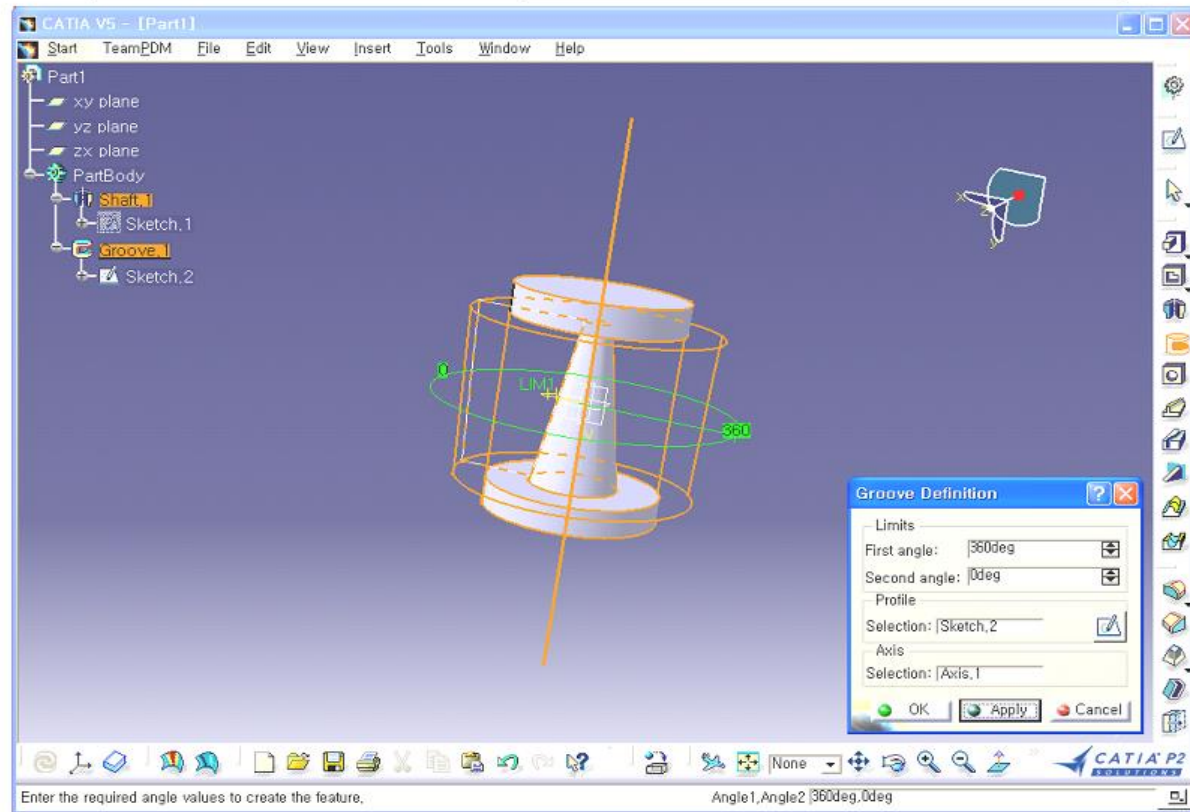
V. PART DESIGN

3-6. Groove




GROOVE

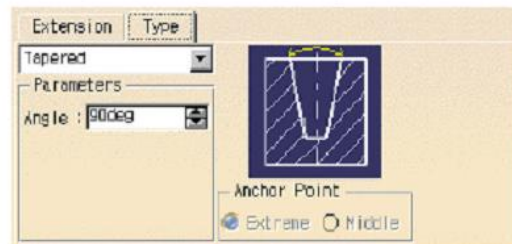
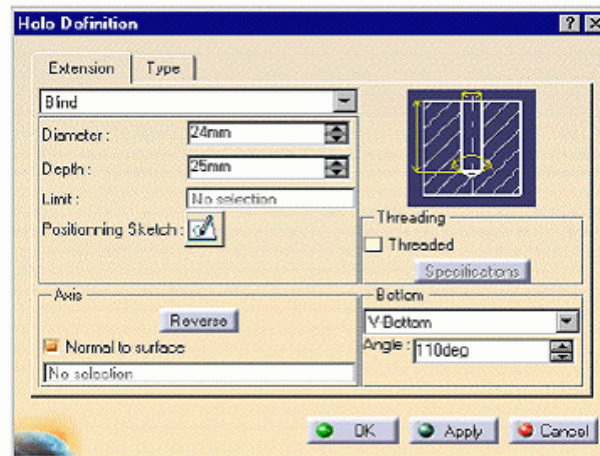
회전형 Solid 형상을 생성하는 기능



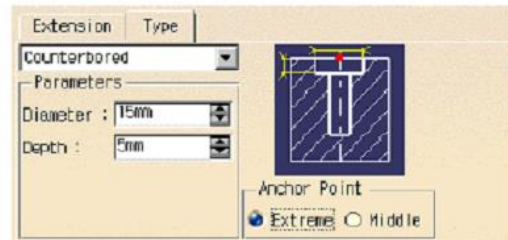
V. PART DESIGN

3-7. Hole (1)

	HOLE	정의되어 있는 형상을 이용하여 구멍 뚫는 작업을 수행
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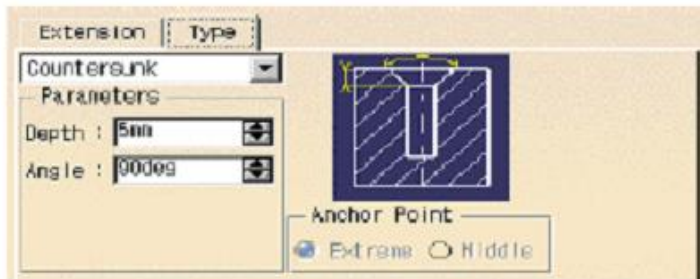
- **Tapered Type**
- Parameter / Anchor Point



- **Counterbored Type**
- Parameter / Anchor Point

V. PART DESIGN

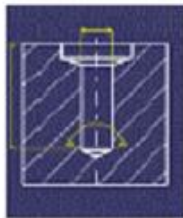
3-7. Hole (2)



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



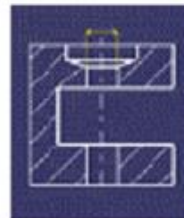
- **Counterdrilled Type**
- Parameter / Anchor Point



Blind



Up to Next



Up to Last



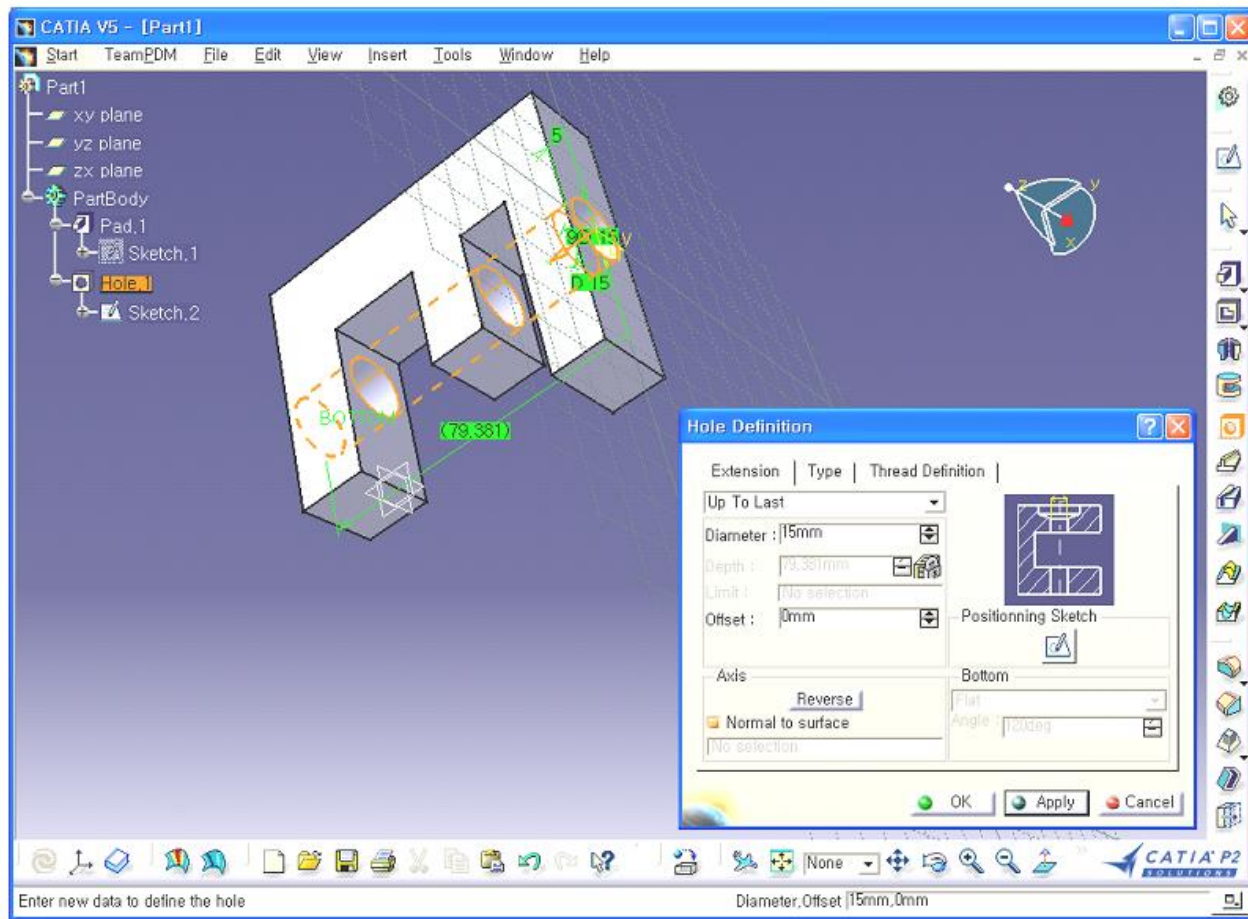
Up to Plane



Up to Surface

V. PART DESIGN

3-7. Hole (3)



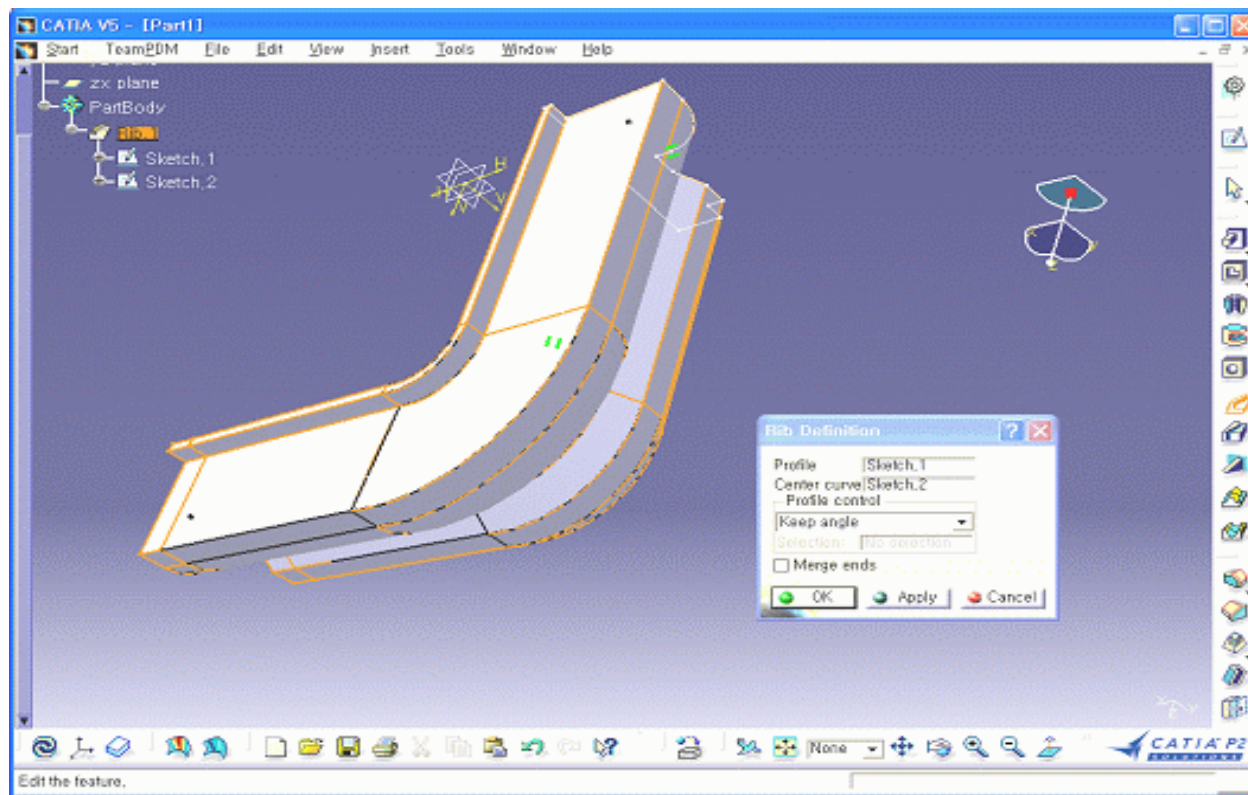
V. PART DESIGN

3-8. Rib



RIB

Profile을 center curve를 따라 가는
solid 형상을 생성하는 기능



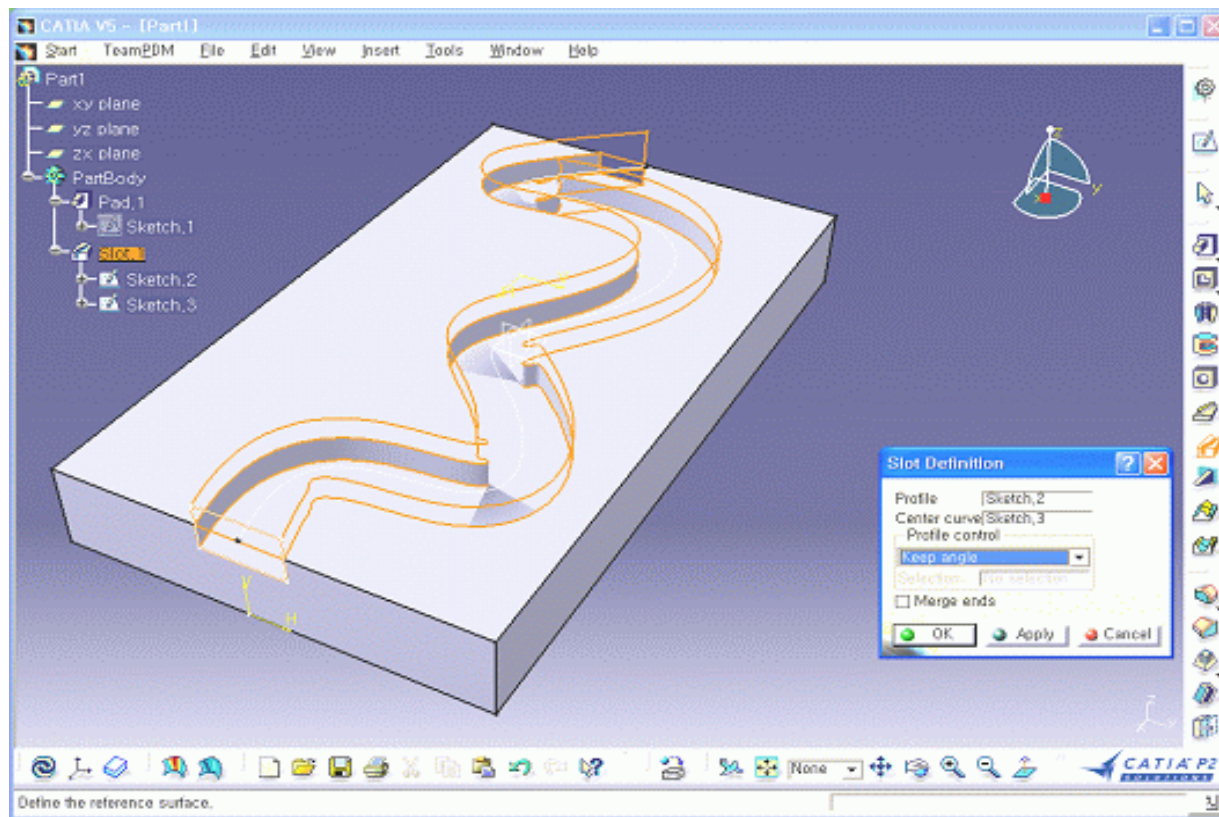
V. PART DESIGN

3-9. Slot



SLOT

Profile을 center curve를 따라 가는
solid 형상을 Pocket하는 기능



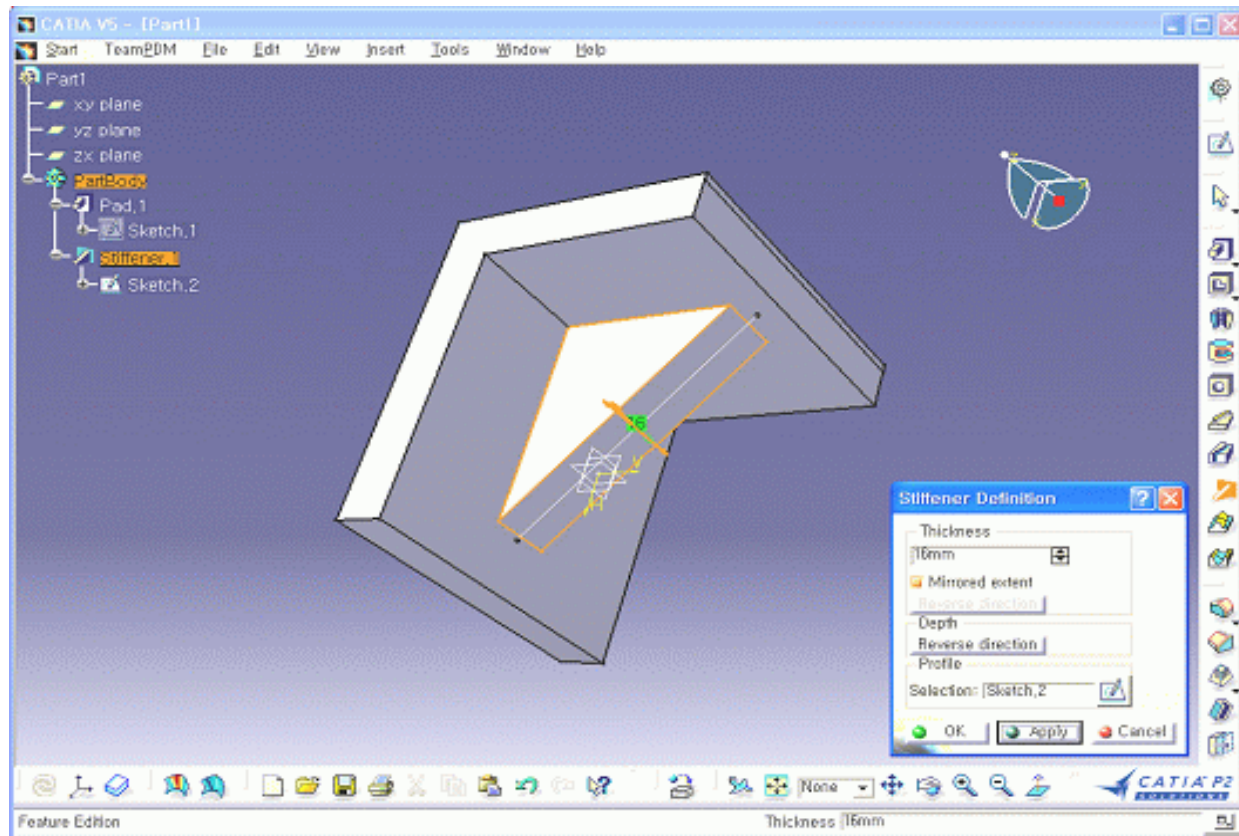
V. PART DESIGN

3-10. Stiffener



STIFFENER

제품의 형상에 보강재를 붙일때



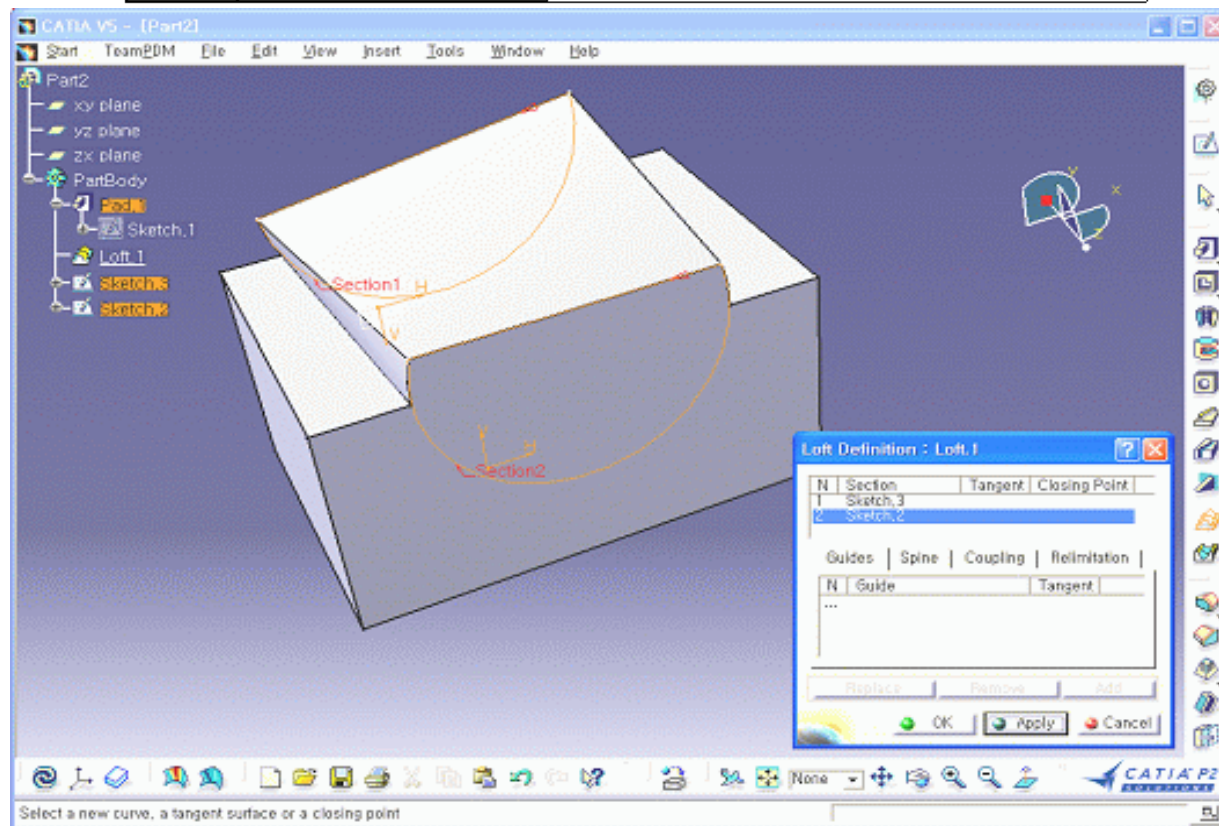
V. PART DESIGN

3-11. Loft




LOFT

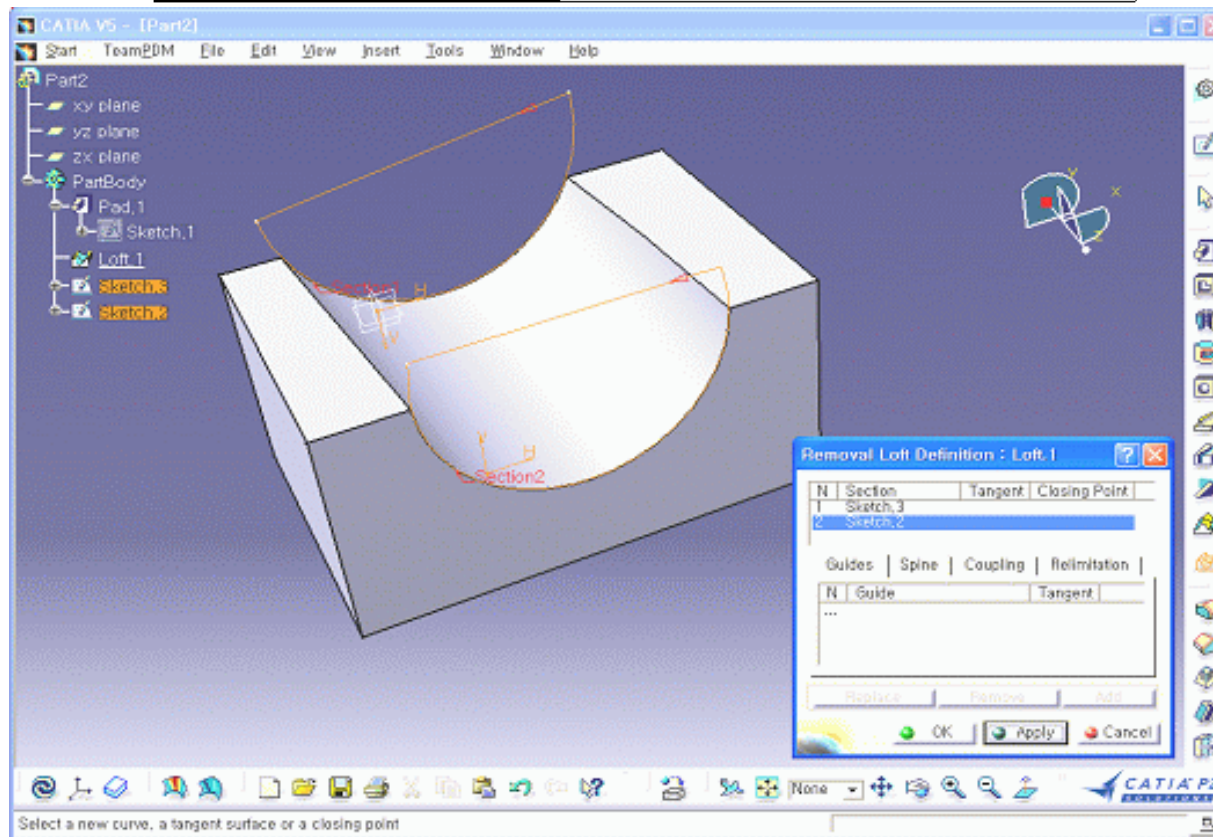
Loft 형 Solid를 생성 (양의 극성)



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



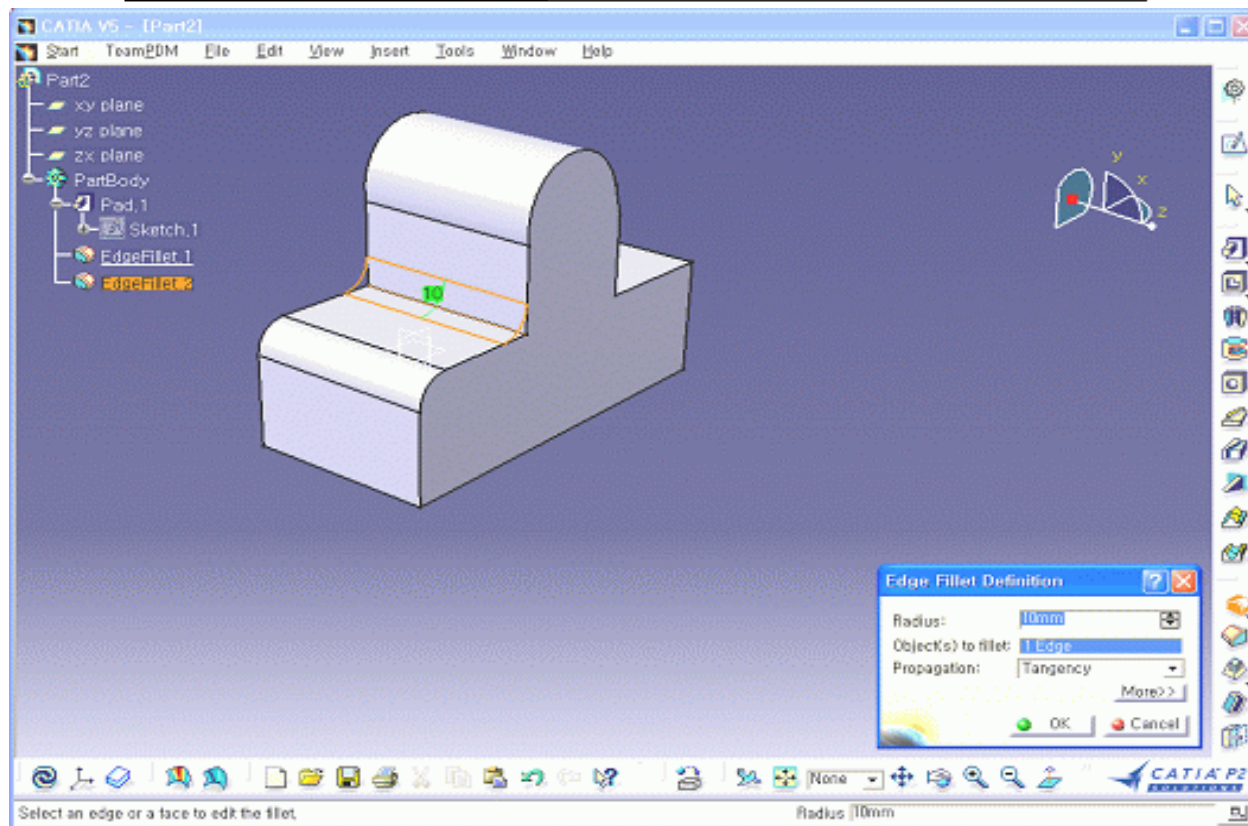
V. PART DESIGN

4-1. Edge Fillet



EDGE FILLET

Edge를 선택하여 라운딩 작업을 수행



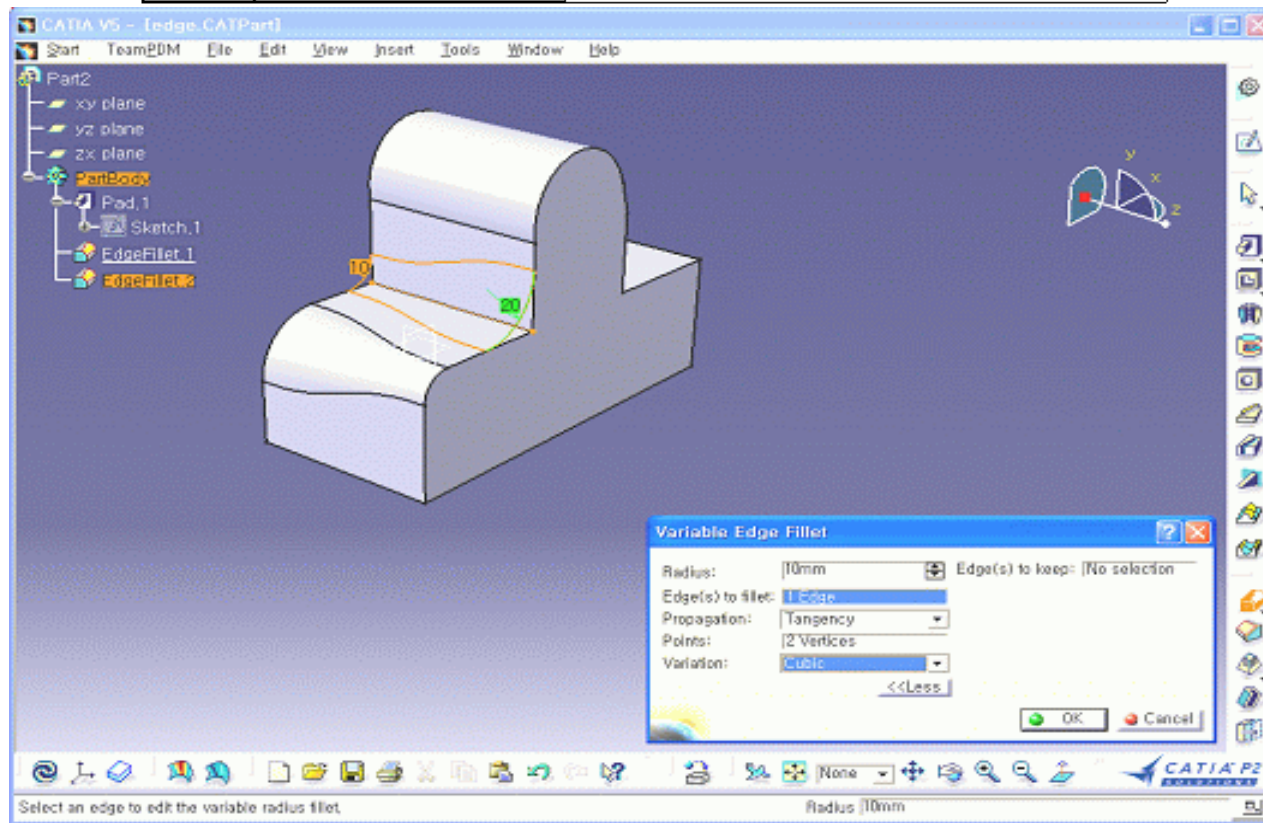
V. PART DESIGN

4-2. Variable Radius Fillet



VARIABLE RADIUS FILLET

반경이 다른 라운딩 작업을 수행



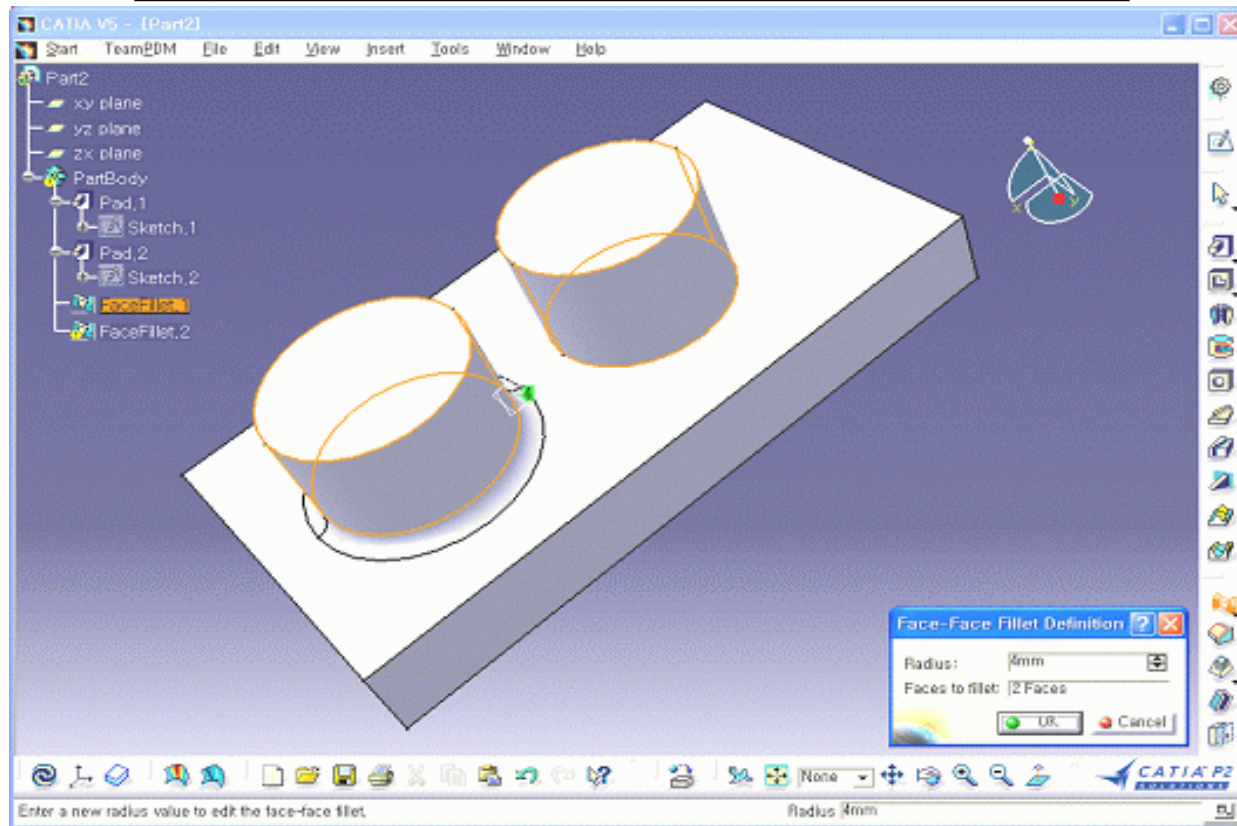
V. PART DESIGN

4-3. Face-Face Fillet



FACE-FACE
FILLET

Face 와 Face 사이에 라운딩 작업



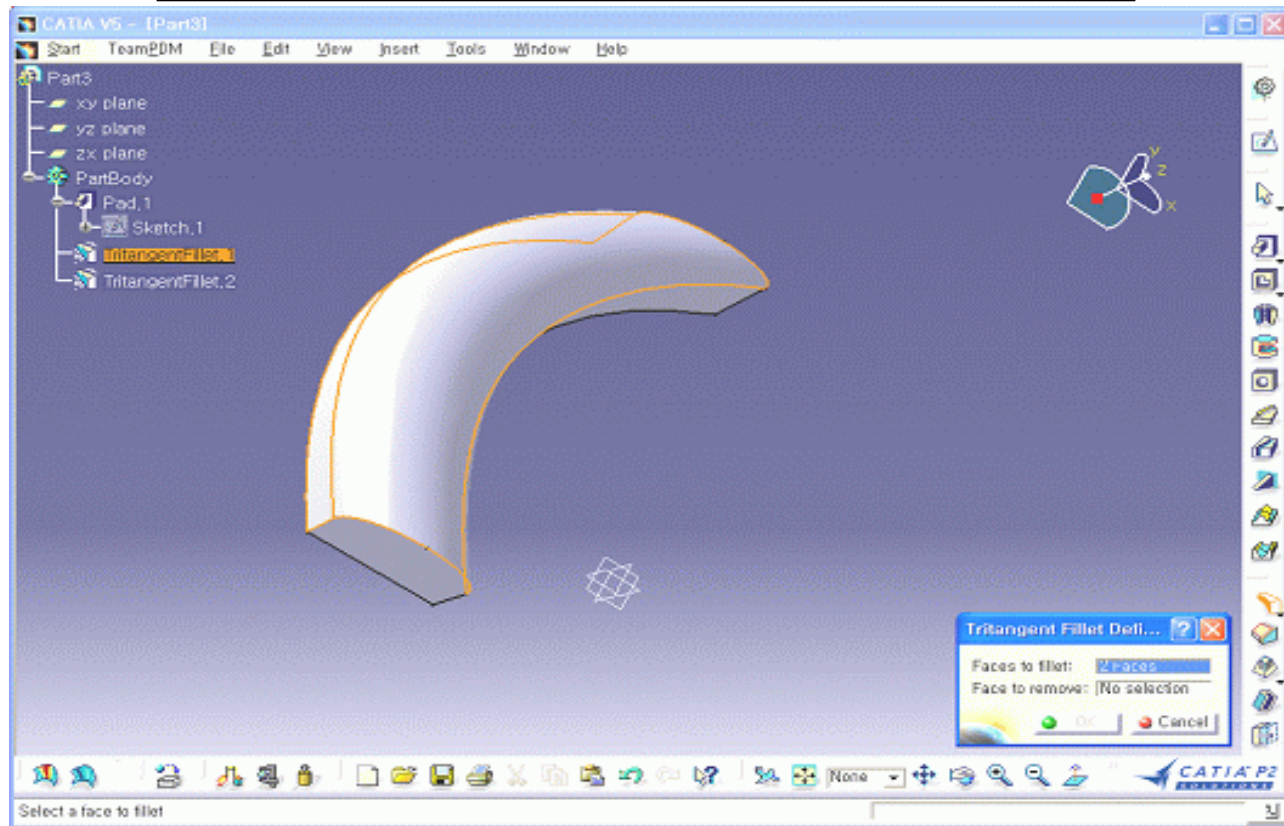
V. PART DESIGN

4-4. Tritangent Fillet



**TRITANGENT
FILLET**

두 면에 대한 Fillet 작업을 수행



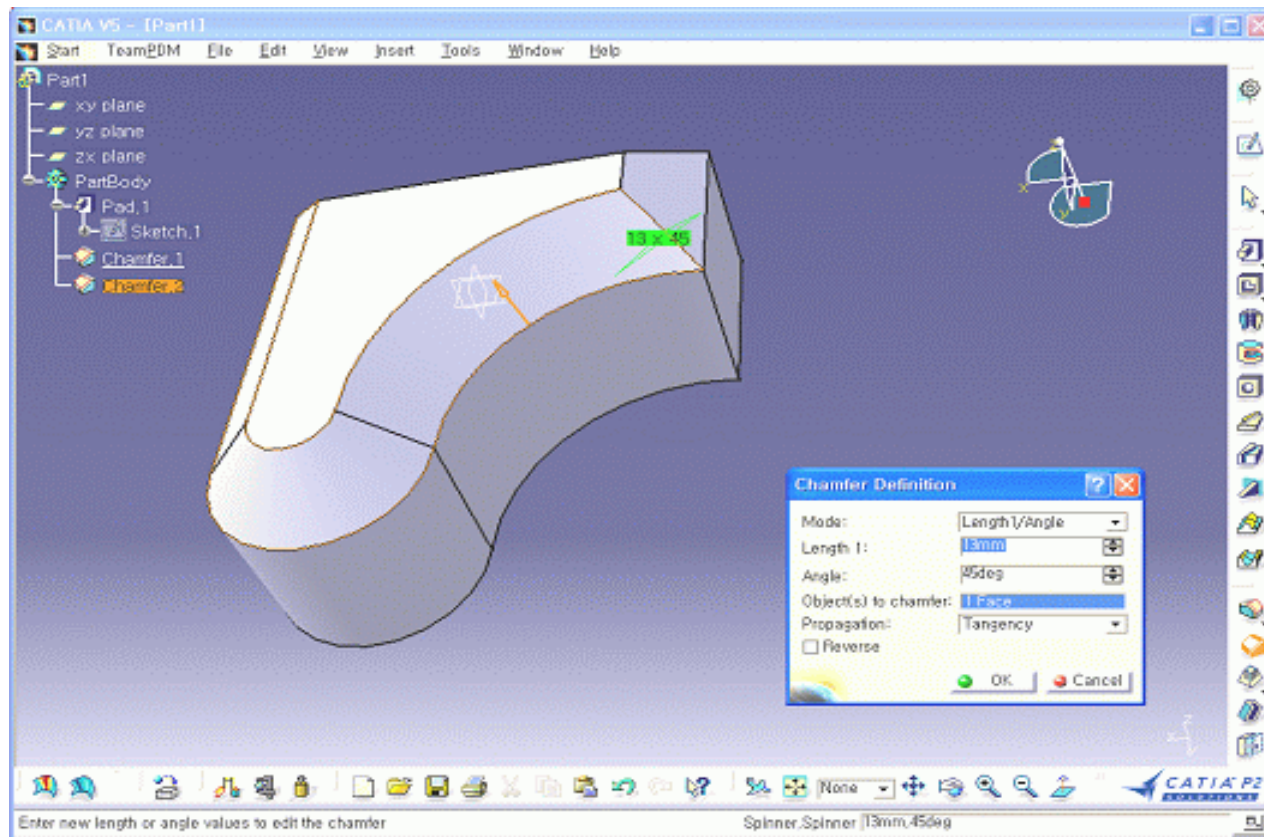
V. PART DESIGN

4-5. Chamber



CHAMBER

모따기 작업을 수행



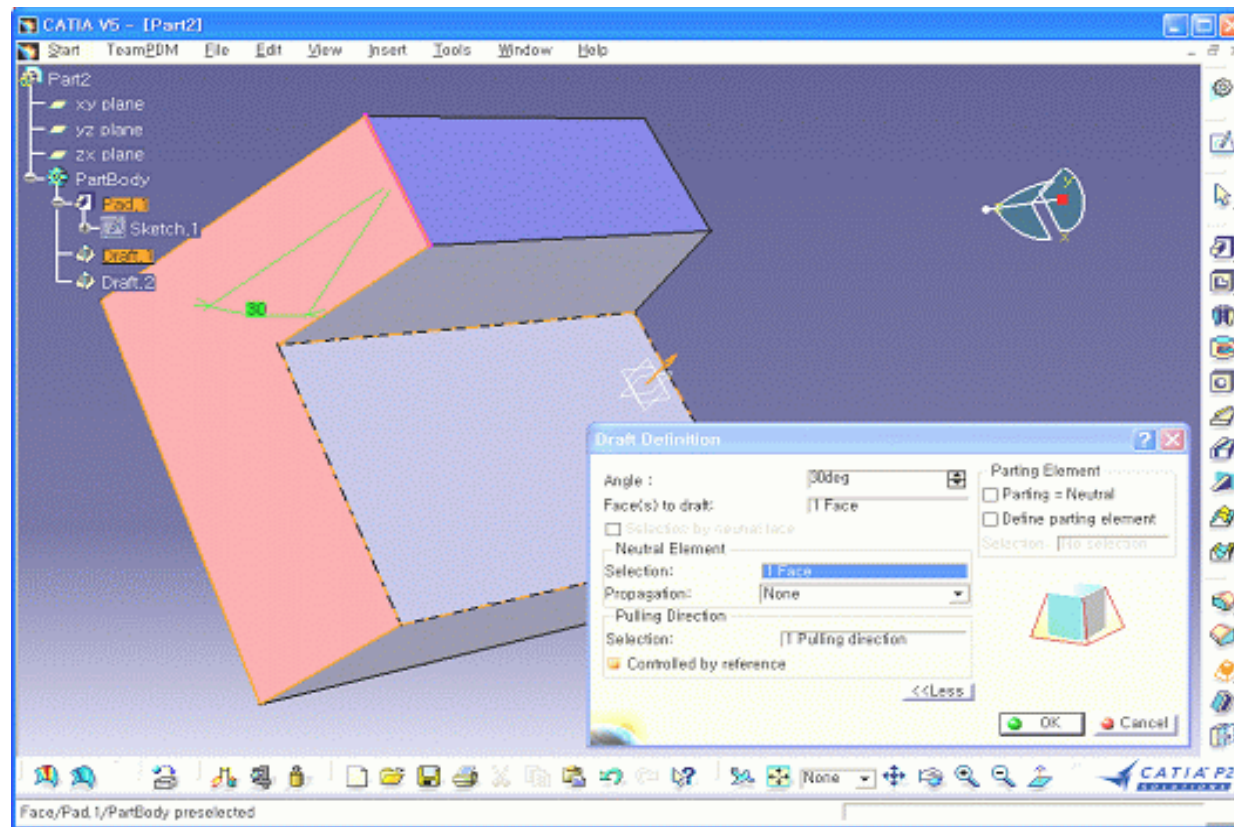
V. PART DESIGN

4-6. Draft



DRAFT

제품형상에 Drafting 작업을 수행



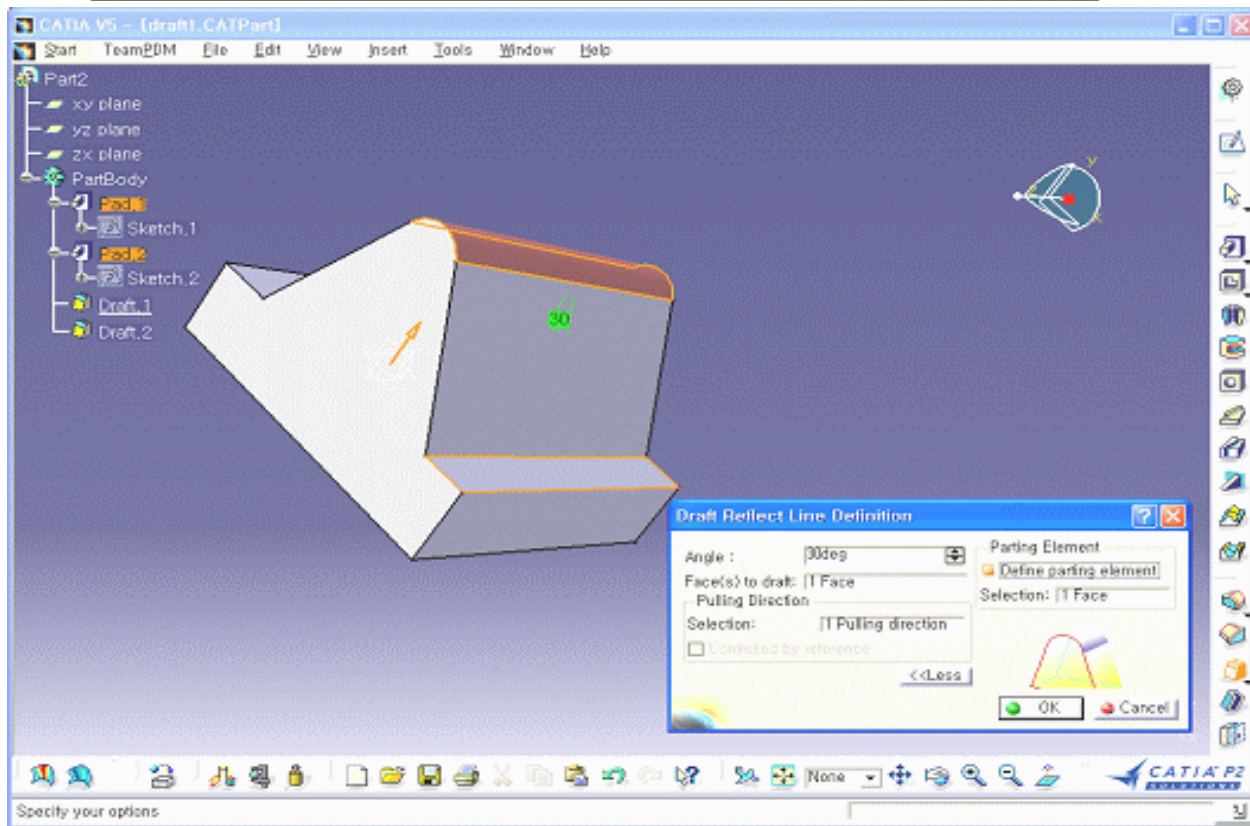
V. PART DESIGN

4-7. Draft Reflect Line




**DRAFT
REFLECT LINE**

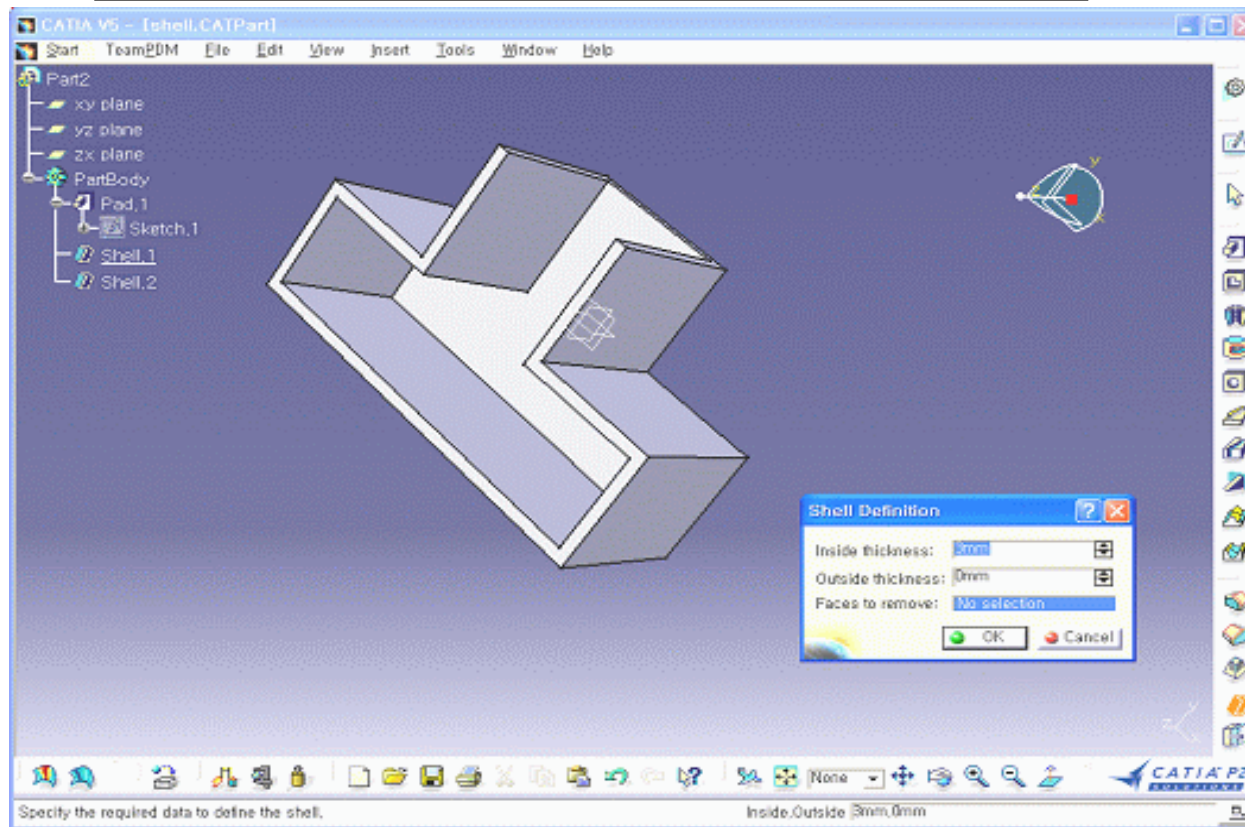
Reflect Line을 이용한 drafting을 수행



V. PART DESIGN

4-8. Shell

	SHELL	제품의 형상에서 일정한 부분을 빼는 작업
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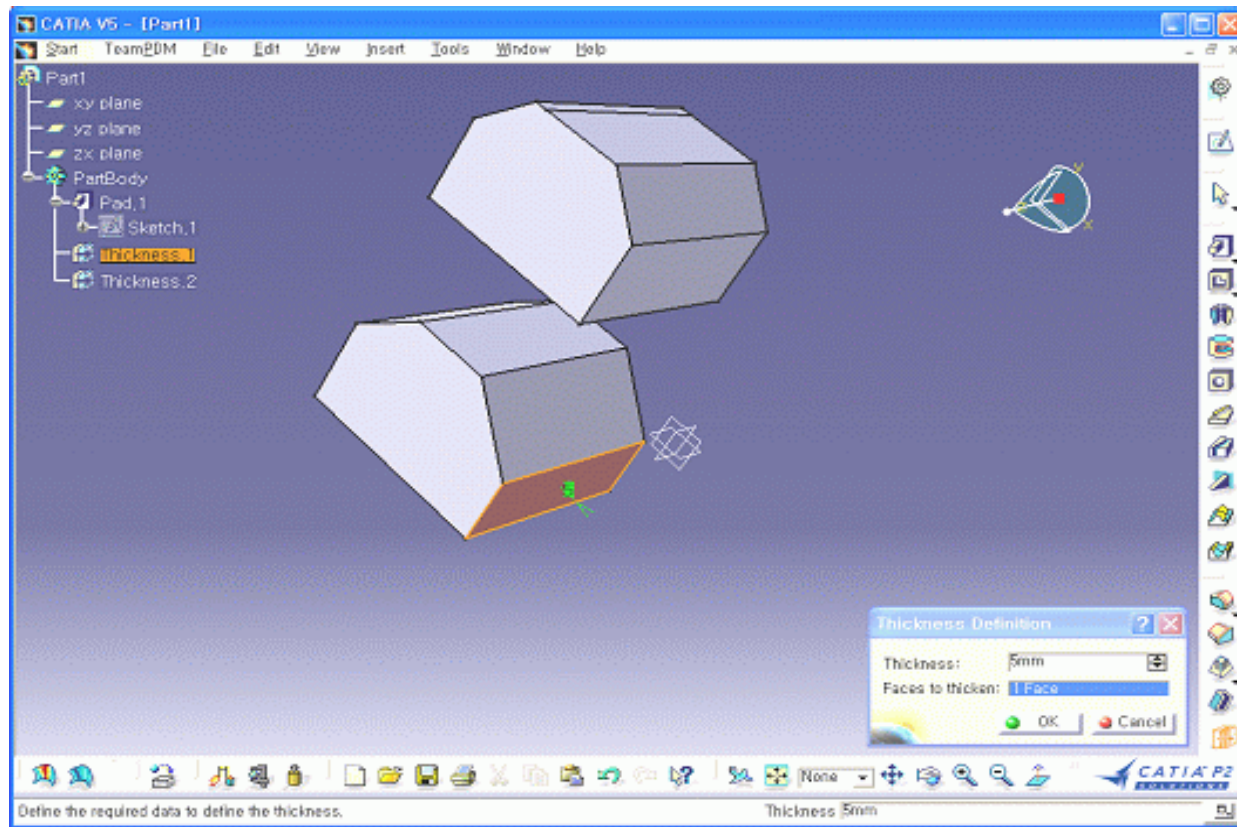
V. PART DESIGN

4-9. Thickness



THICKNESS

제품형상에 사용자가 두께를 늘리고
자 할때 사용하는 기능



V. PART DESIGN

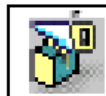
5. Surface-Based Features

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



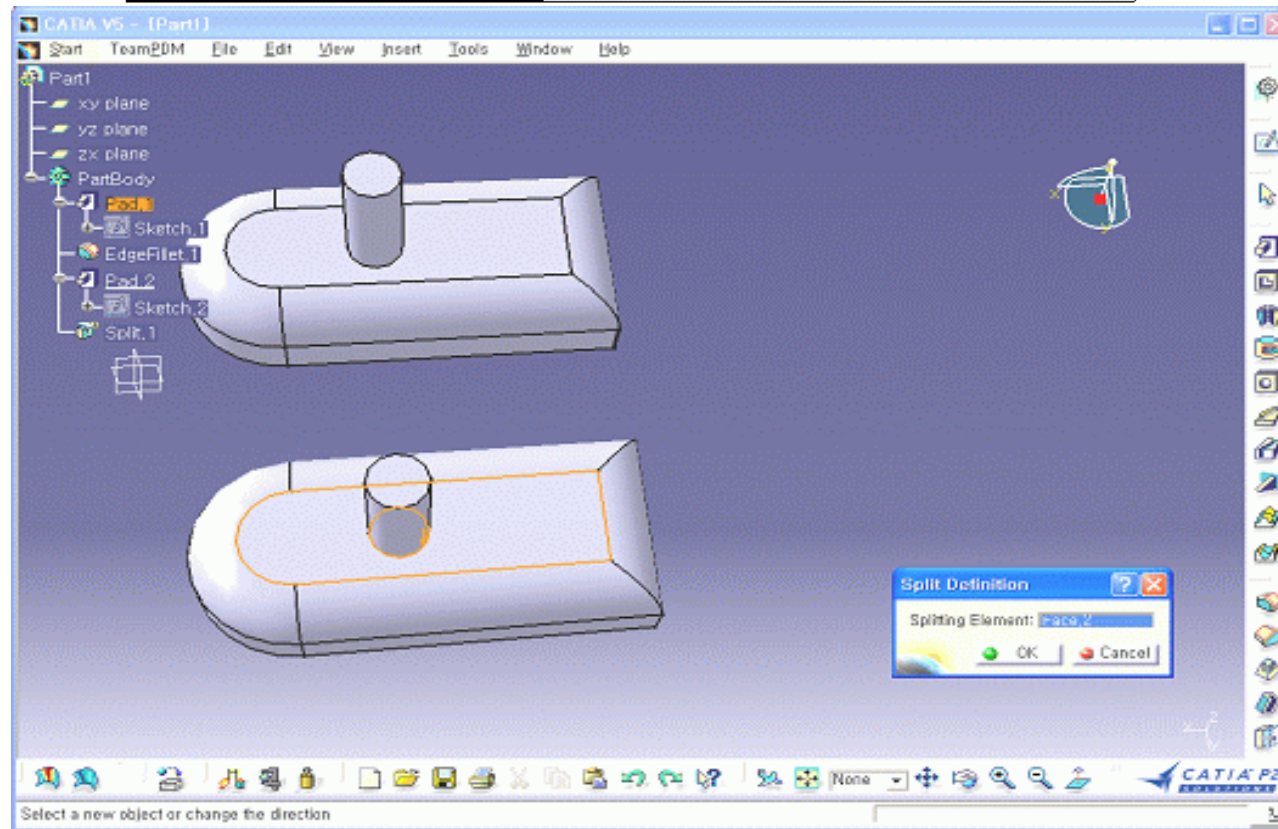
V. PART DESIGN

5-1. Split



SPLIT

형상의 일부분을 임의 지점까지 잘라
낼 때 사용하는 기능



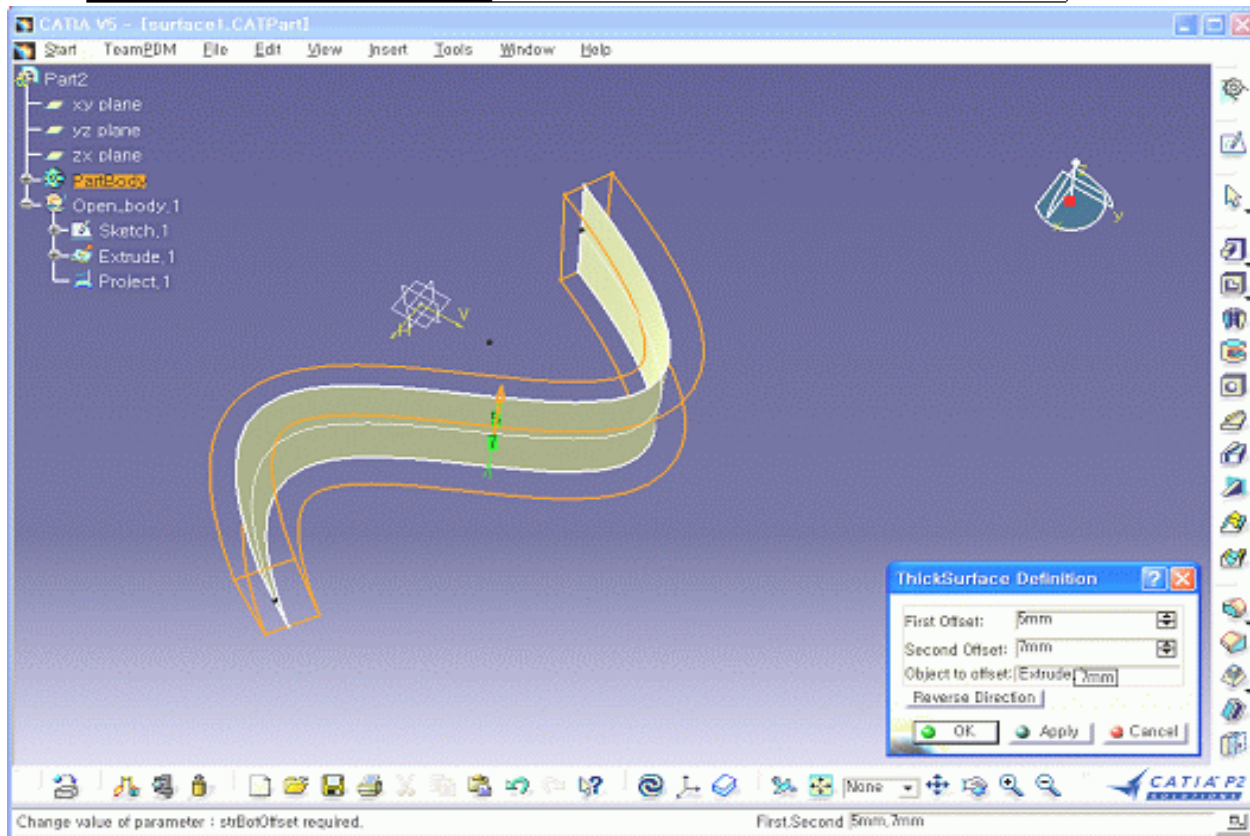
V. PART DESIGN

5-2. Thick Surface



THICK SURFACE

실제 surface를 이용하여 그 surface
에 두께를 부여하면서 Solid 생성



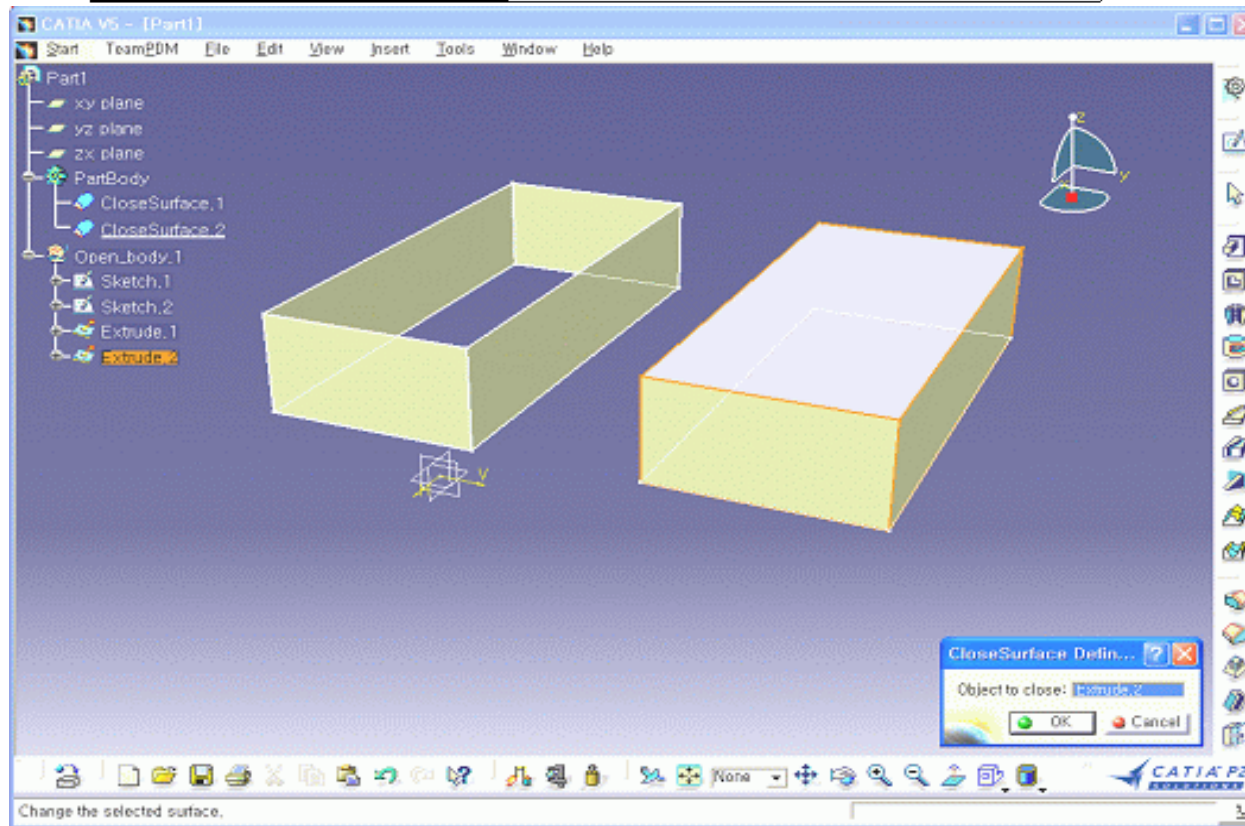
V. PART DESIGN

5-3. Close Surface



CLOSE
SURFACE

Surface를 솔리드로 채워주는 기능



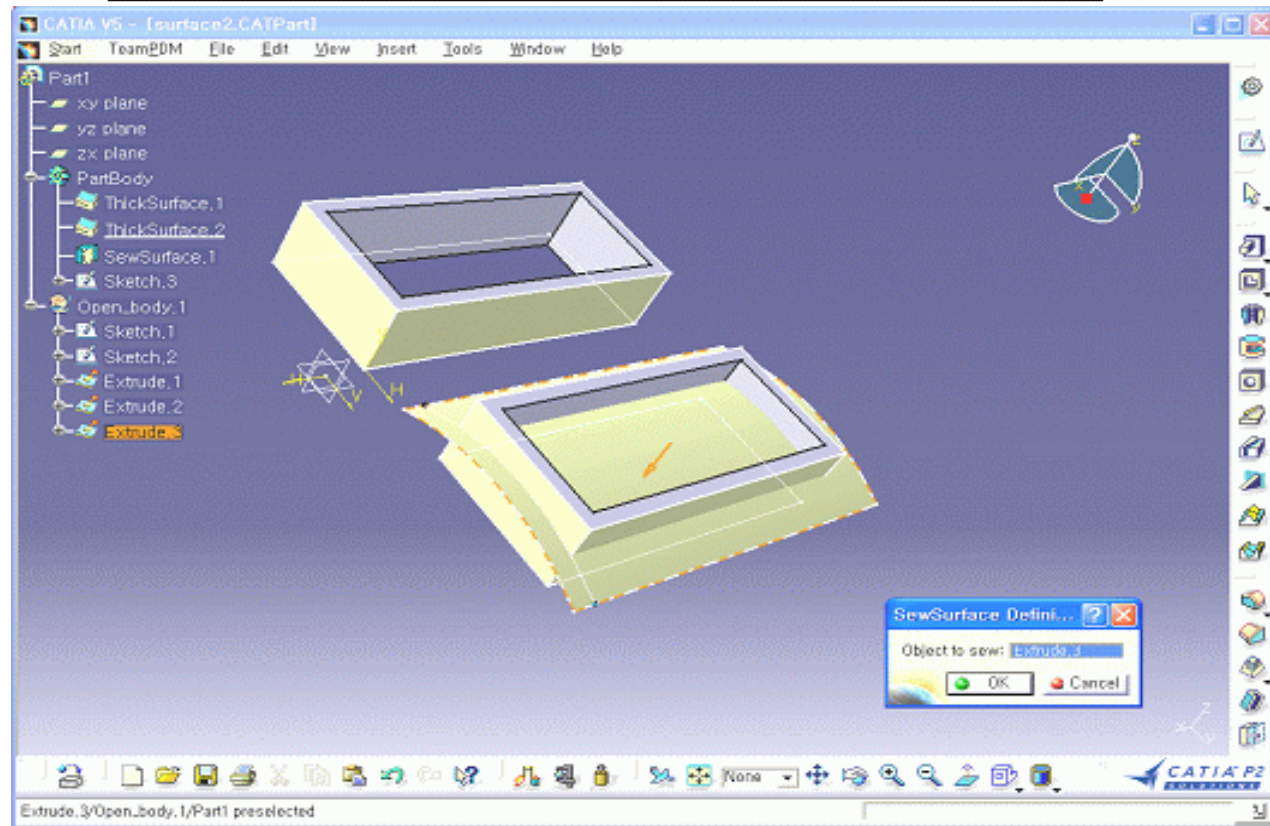
V. PART DESIGN

5-4. Sew Surface



SEW SURFACE

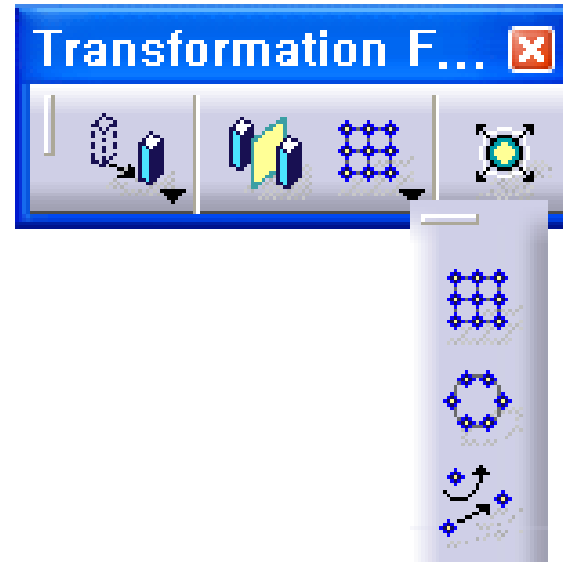
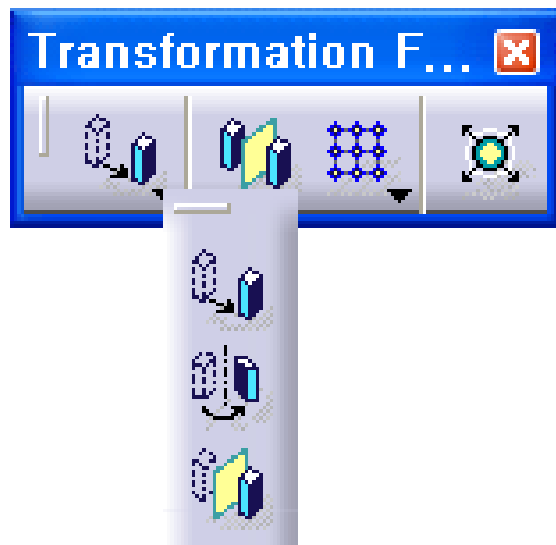
Surface를 솔리드에 붙여는 기능



V. PART DESIGN

6. Transformation Features

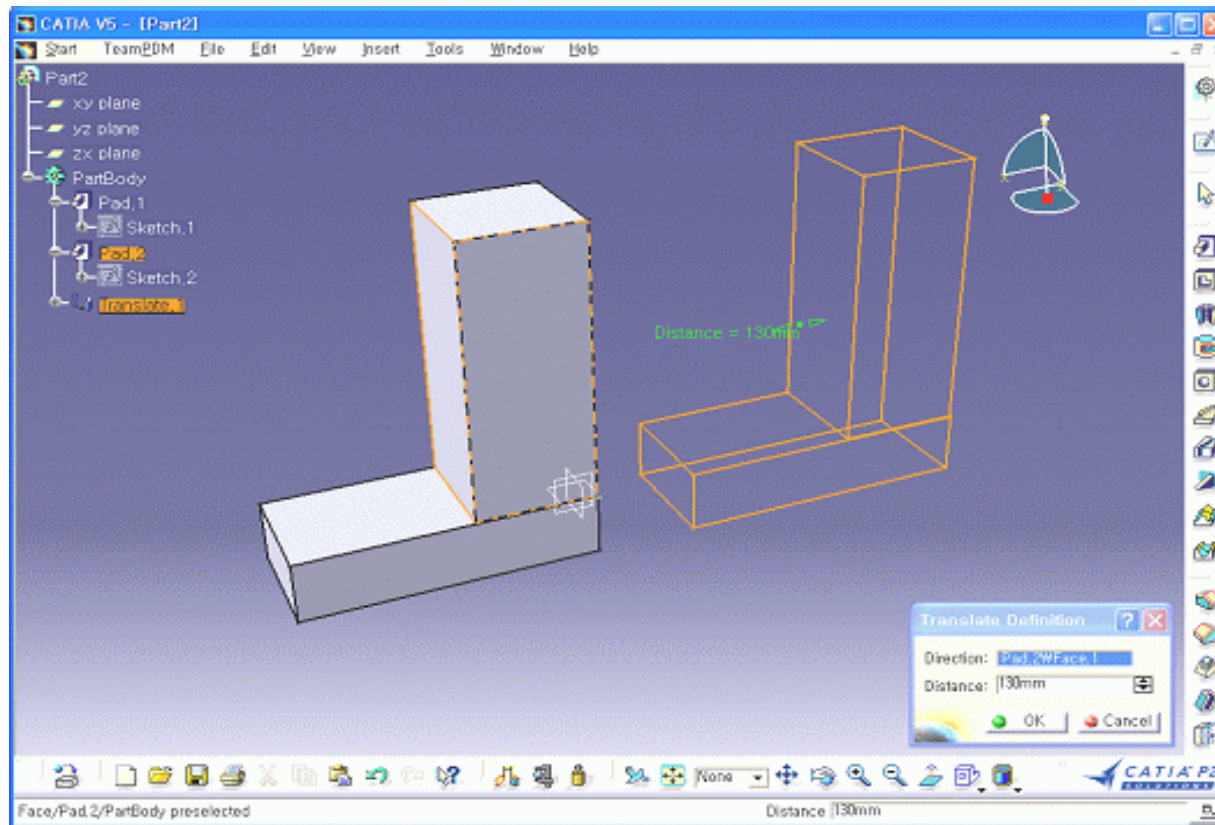
솔리드의 이동이나 복사등의 아이콘을 모아놓은 툴바로 Feature 들을 변형 할 때 사용하는 기능들이 조합



V. PART DESIGN

6-1. Translation

	TRANSLATION	생성되어 있는 Solid 자체를 이동
--	--------------------	----------------------



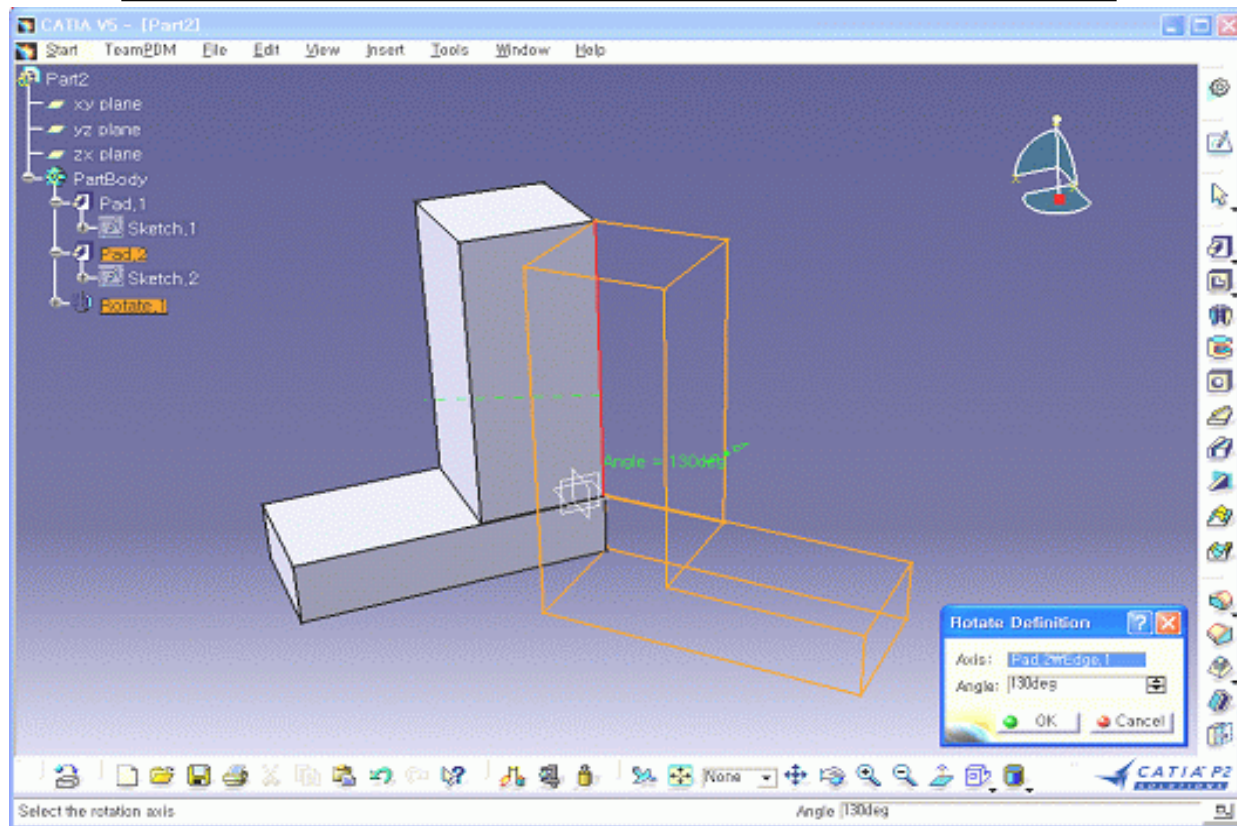
V. PART DESIGN

6-2. Rotation



ROTATION

생성되어 있는 Solid 자체를 회전



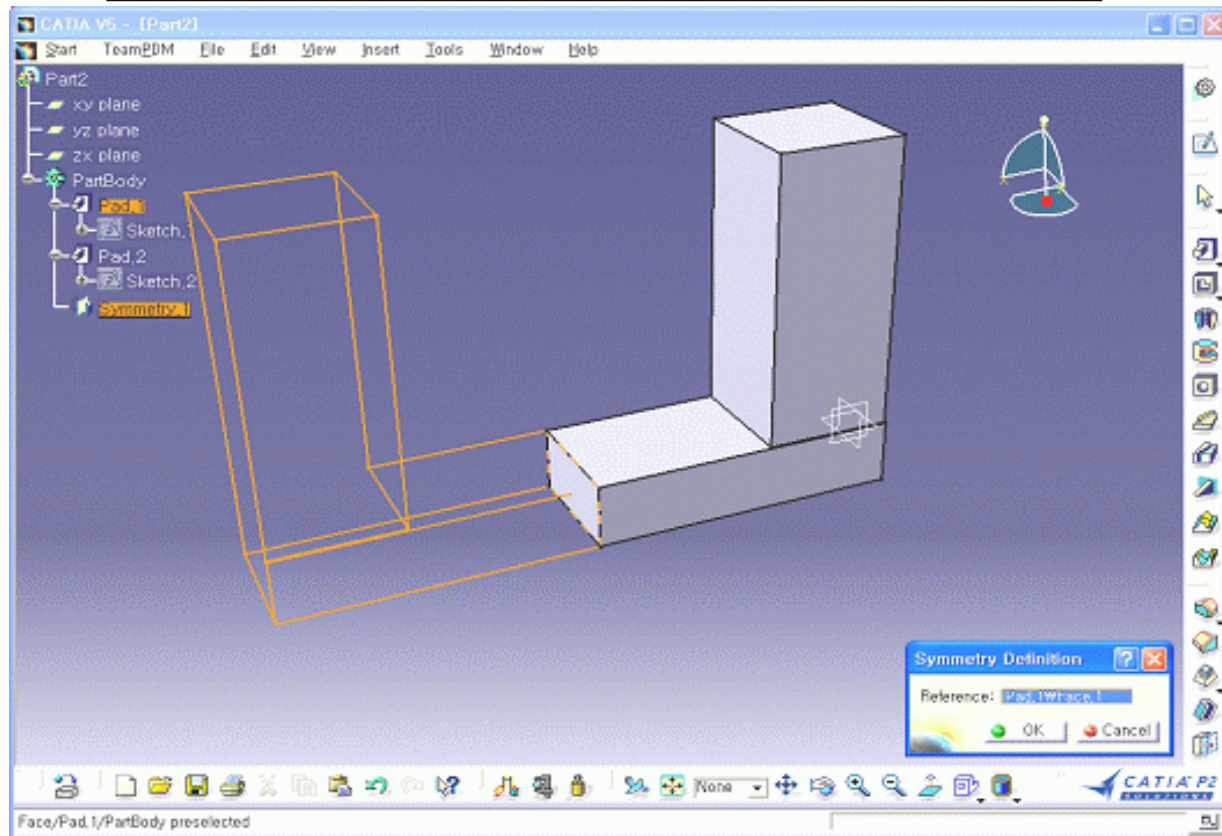
V. PART DESIGN

6-3. Symmetry



SYMMETRY

생성되어 있는 Solid 자체를 대칭이동



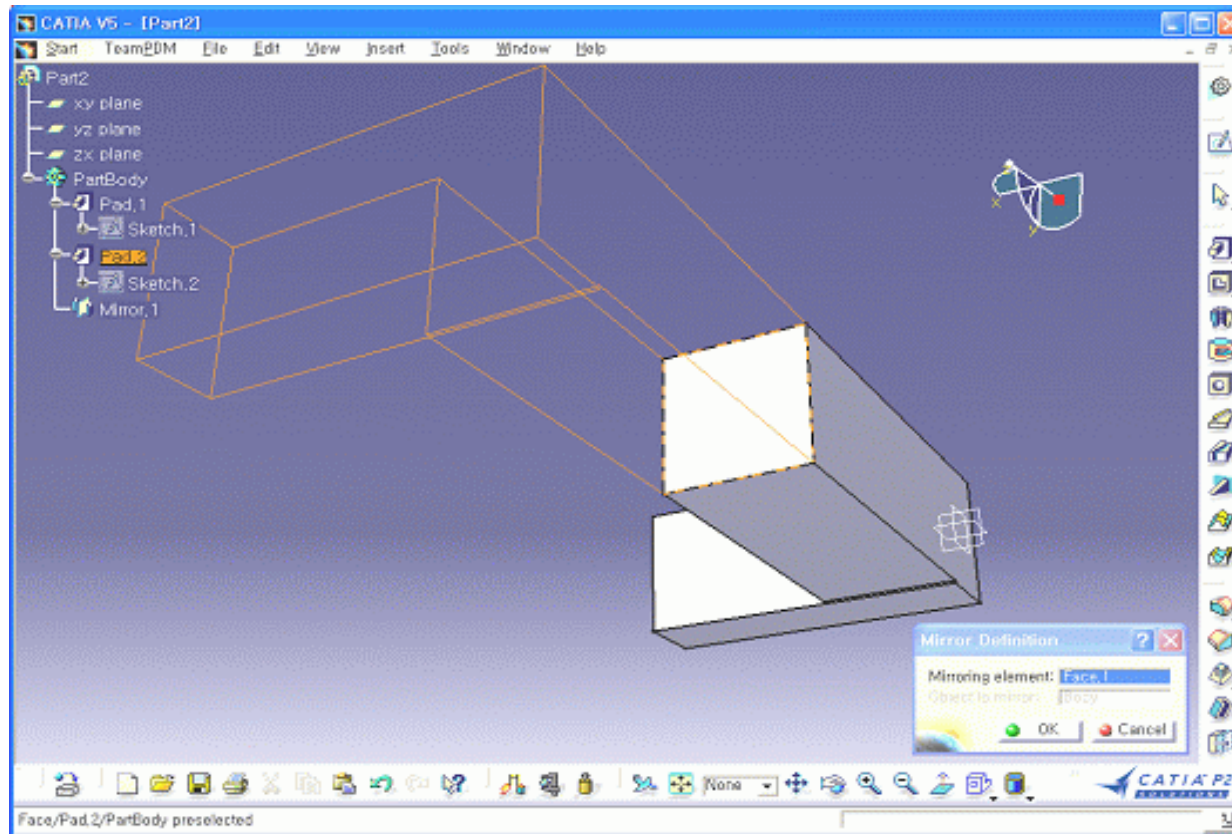
V. PART DESIGN

6-4. Mirror




MIRROR

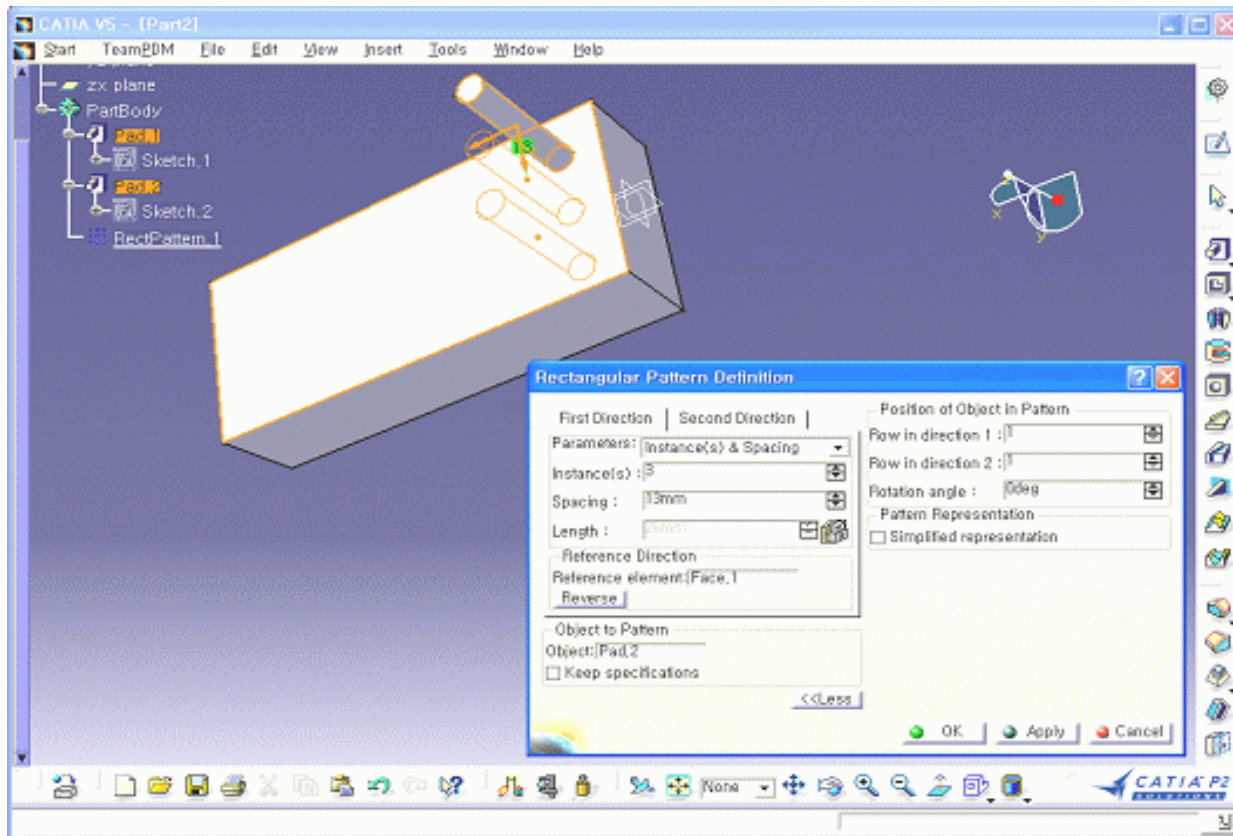
똑같은 형상을 반대편에도 같은 모양으로 만들 때 사용



V. PART DESIGN

6-5. Rectangular Pattern

	RECTANGULAR PATTERN	바둑판 모양으로 여러개의 body를 copy 형태로 같은 모양으로 생성
---	--------------------------------	--



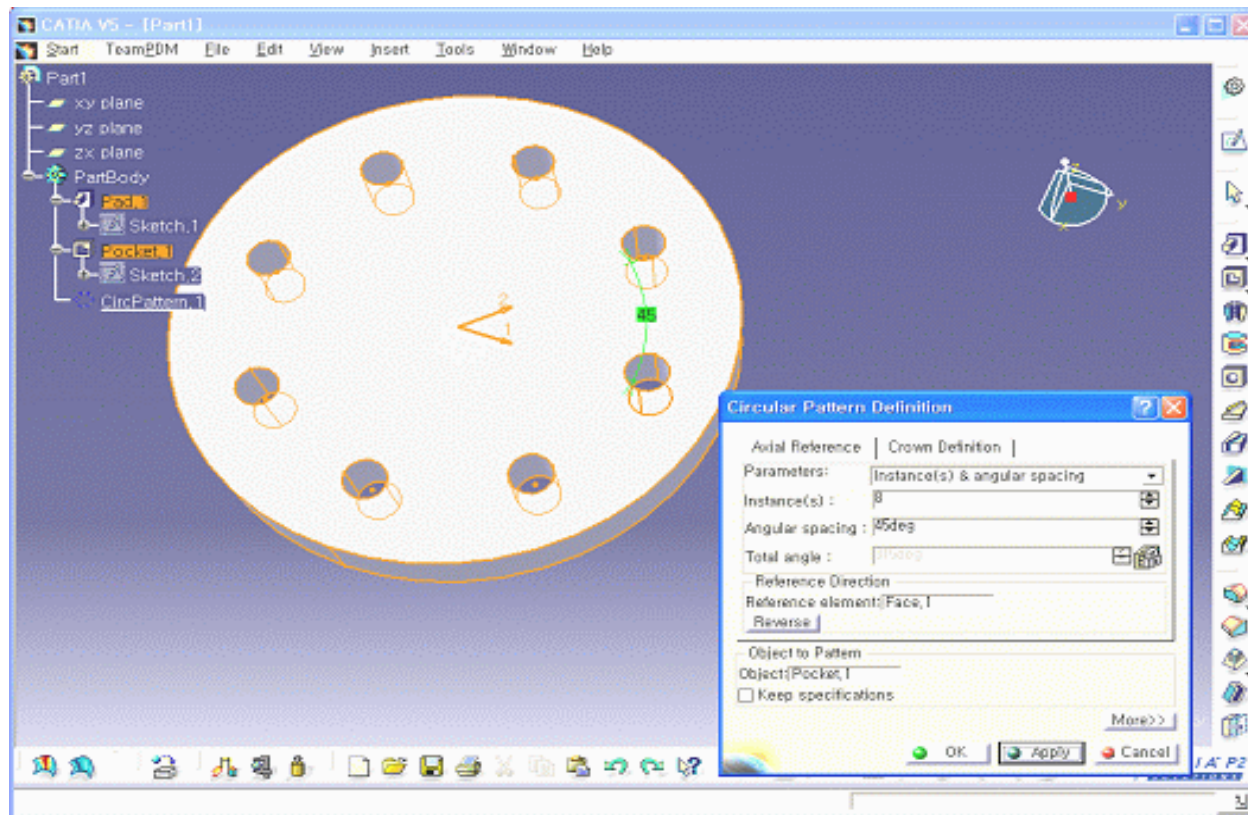
V. PART DESIGN

6-6. Circular Pattern



CIRCULAR PATTERN

원형 모양으로 여러개의 body를 copy
형태로 같은 모양으로 생성



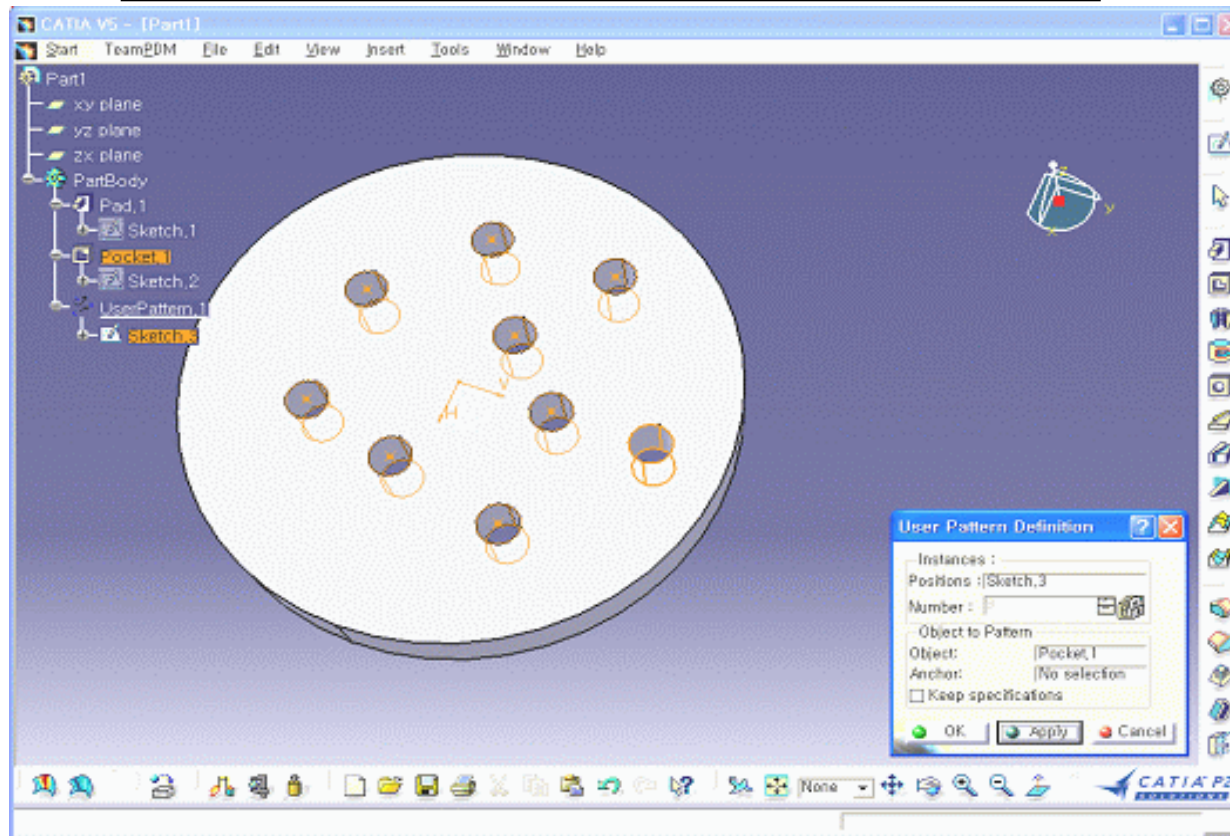
V. PART DESIGN

6-7. User Pattern



USER PATTERN

사용자가 정의한 형태로 Pattern 처리



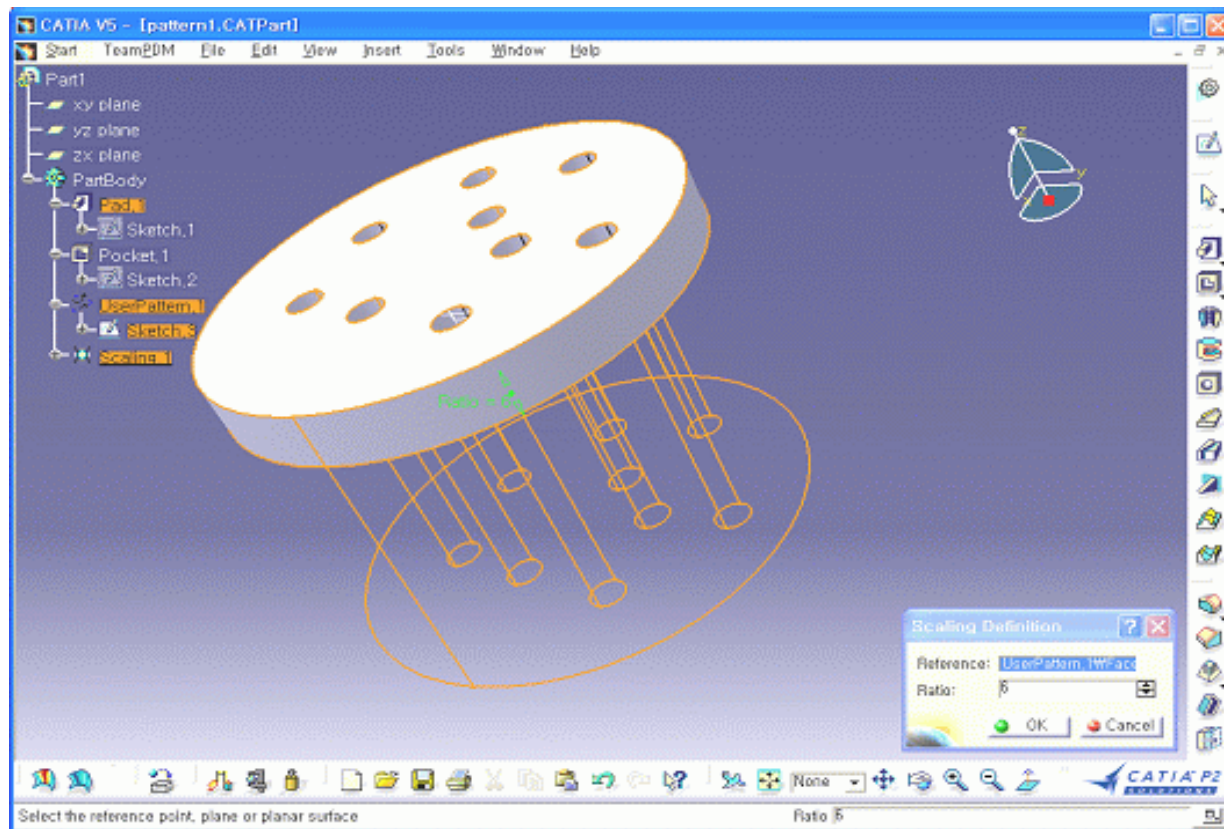
V. PART DESIGN

6-8. Scaling



SCALING

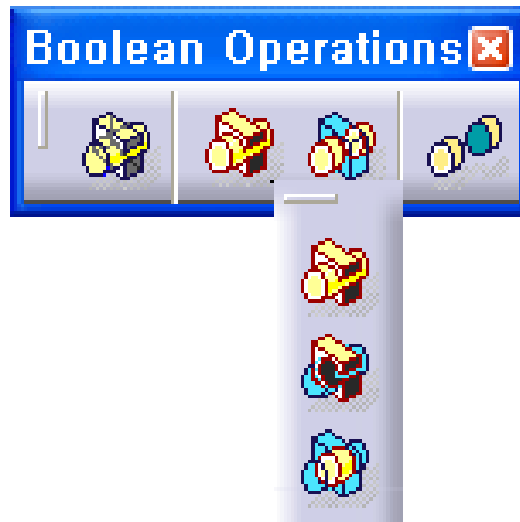
형상의 크기를 조절하는 기능



V. PART DESIGN

7. Boolean Operations

Body 와 Body를 Boolean 연산을 해주는 아이콘을 모아놓은 툴 바



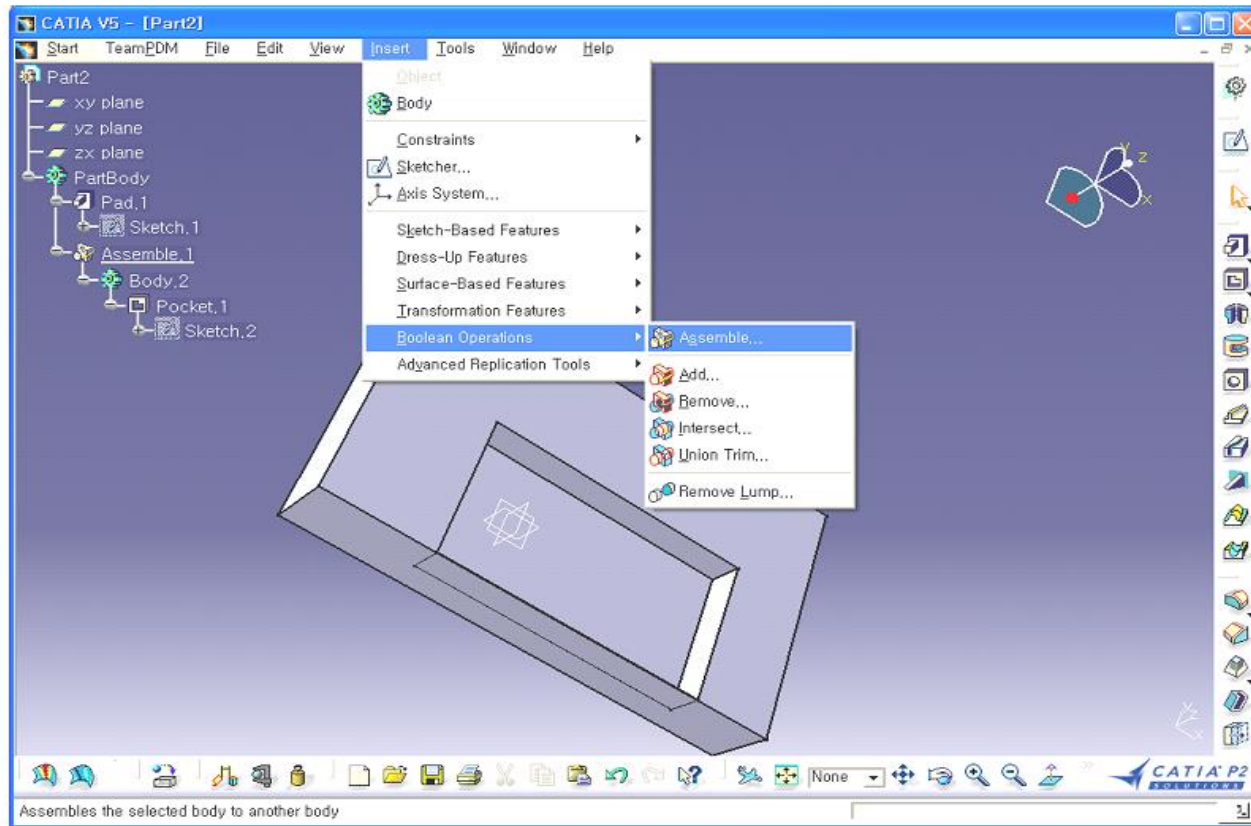
V. PART DESIGN

7-1. Assemble




ASSEMBLE

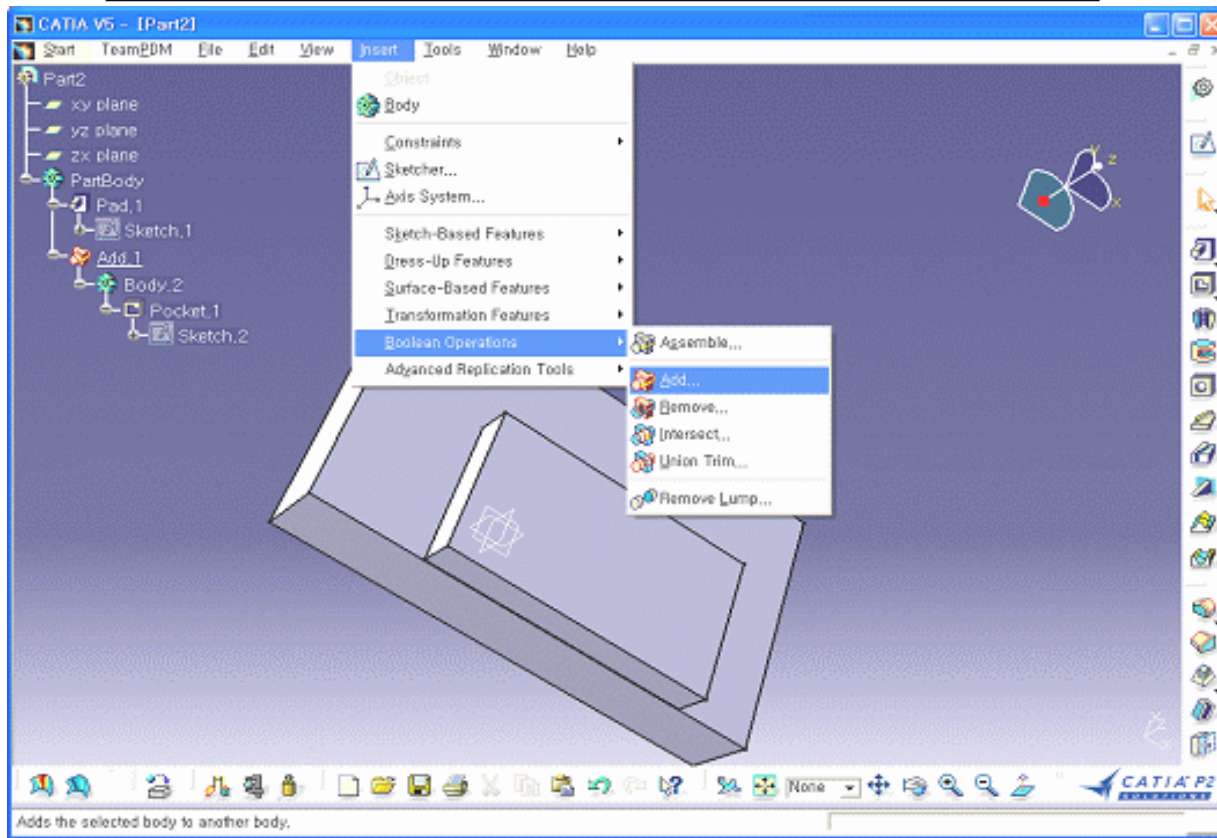
극성을 유지하면서 Union 작업을 수행



V. PART DESIGN


7-2. Add

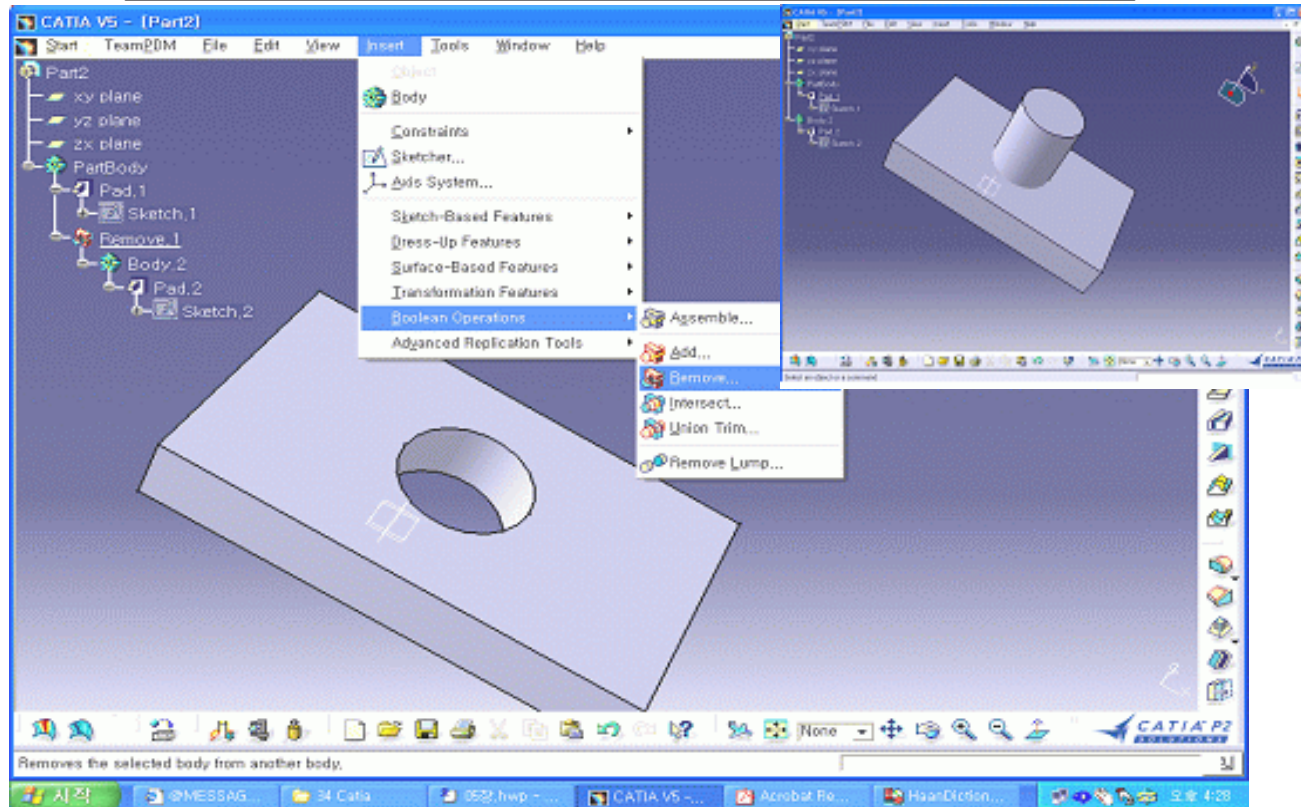
	ADD	+극성을 유지하면서 Union 작업을 수행
---	-----	-------------------------



V. PART DESIGN

7-3. Remove

	REMOVE	Subtract 작업을 수행
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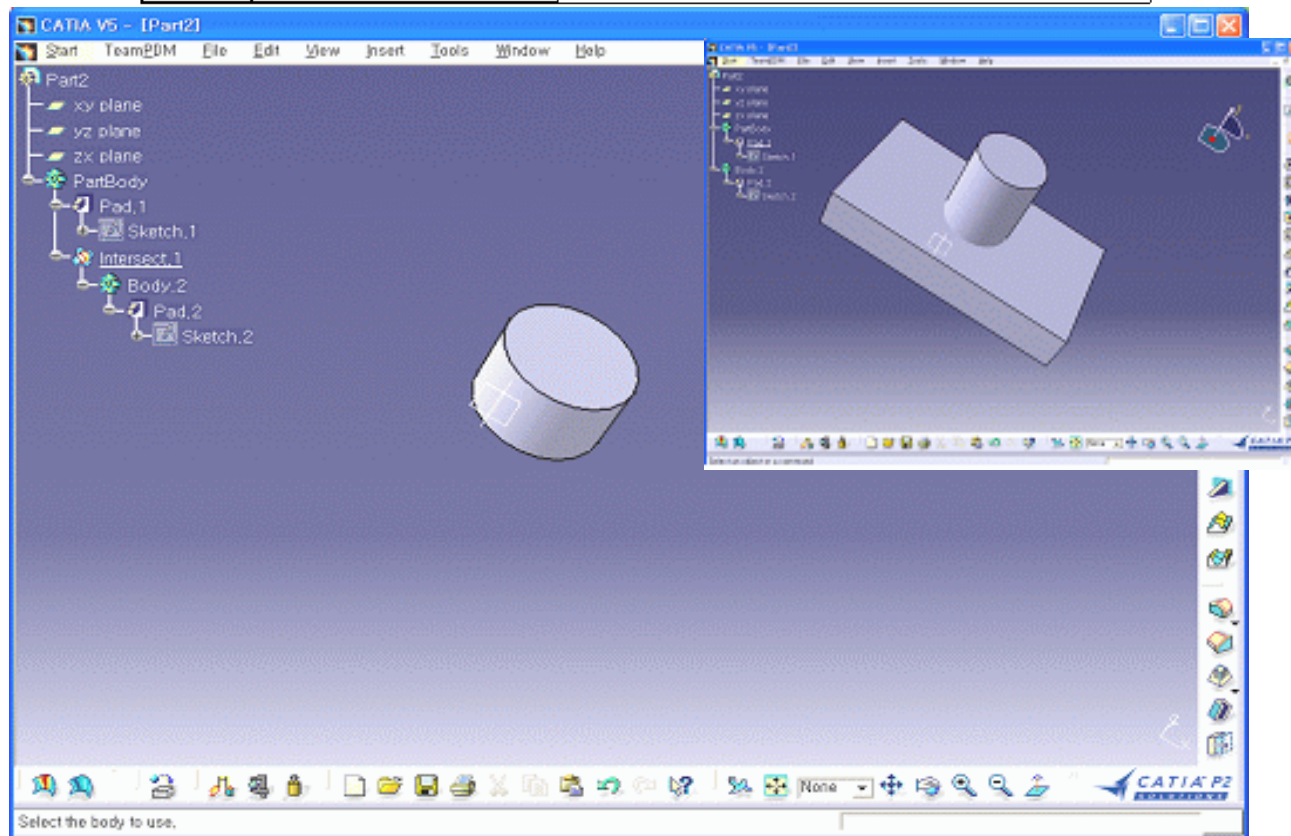
V. PART DESIGN

7-4. Intersect



INTERSECT

서로 교차되는 부분만을 남겨 생성



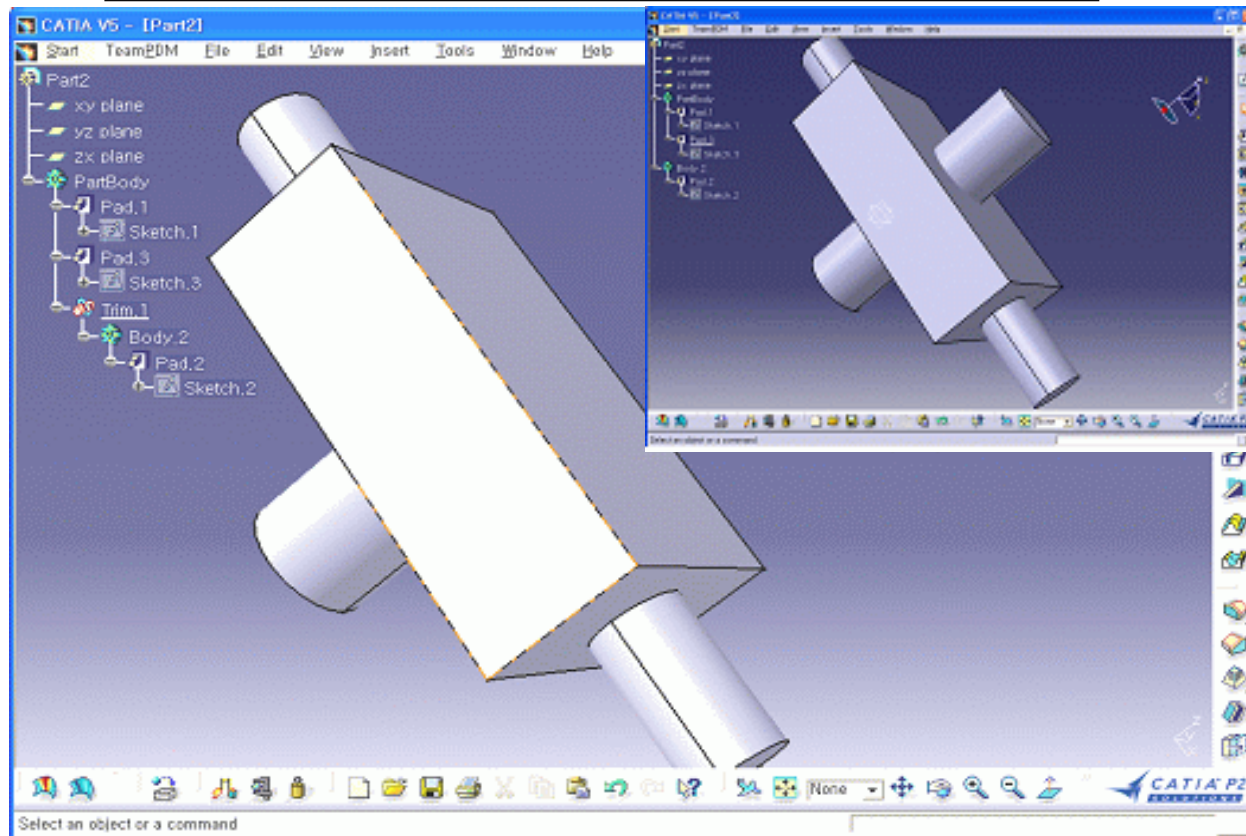
V. PART DESIGN

7-5. Union Trim



UNION TRIM

Union 작업시 trim작업을 동시에 수행



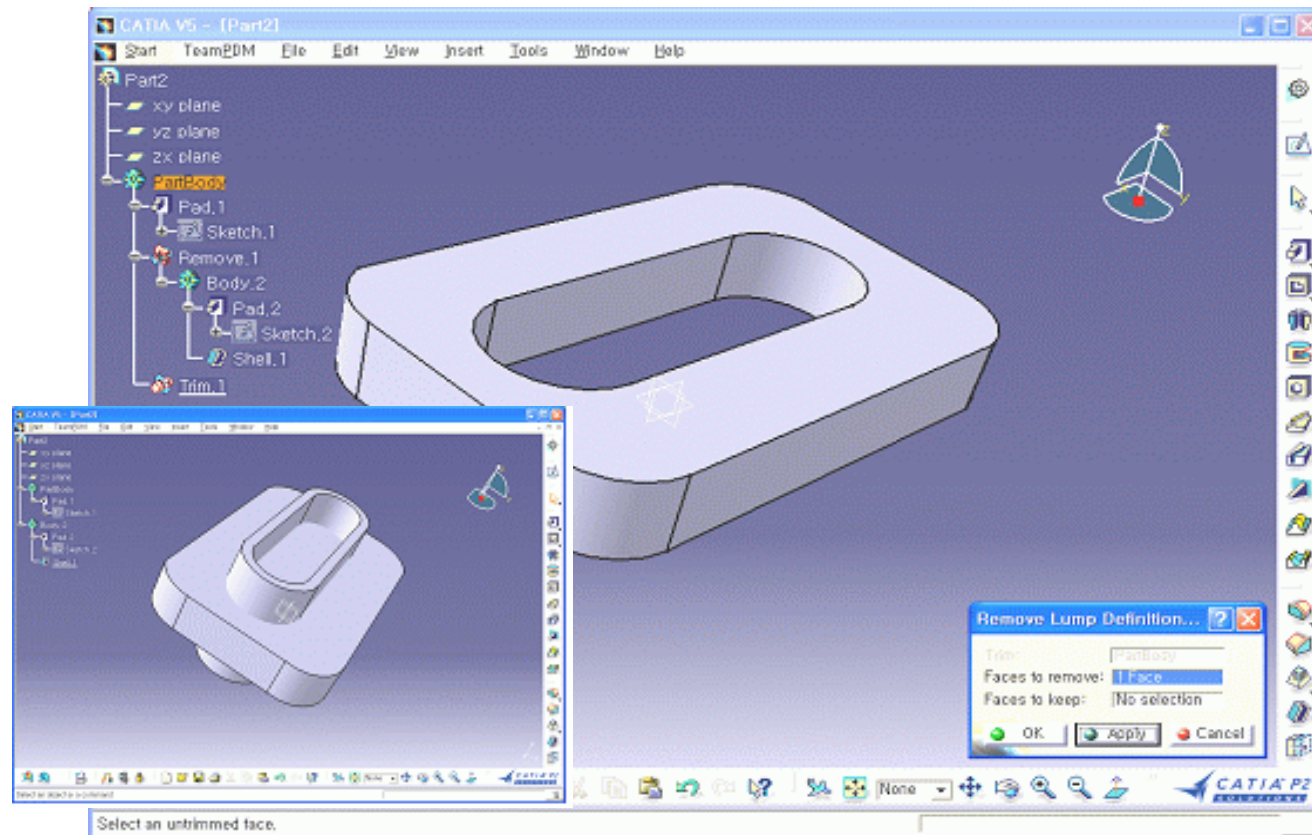
V. PART DESIGN

7-6. Remove Lump



REMOVE LUMP

Remove 작업시 trim작업을 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
---	--------------------	------------------

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
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10-3. Open Catalog

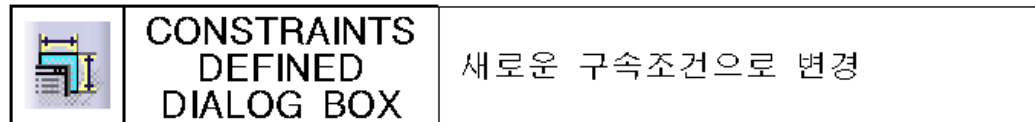
	OPEN CATALOG	카탈로그 열기
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V. PART DESIGN

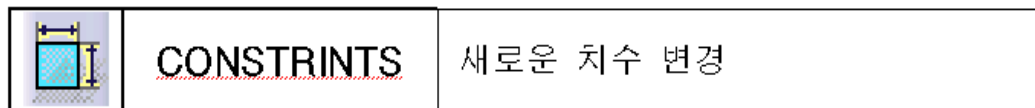
11. Constraints

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

11-1. Constraints Defined Dialog Box

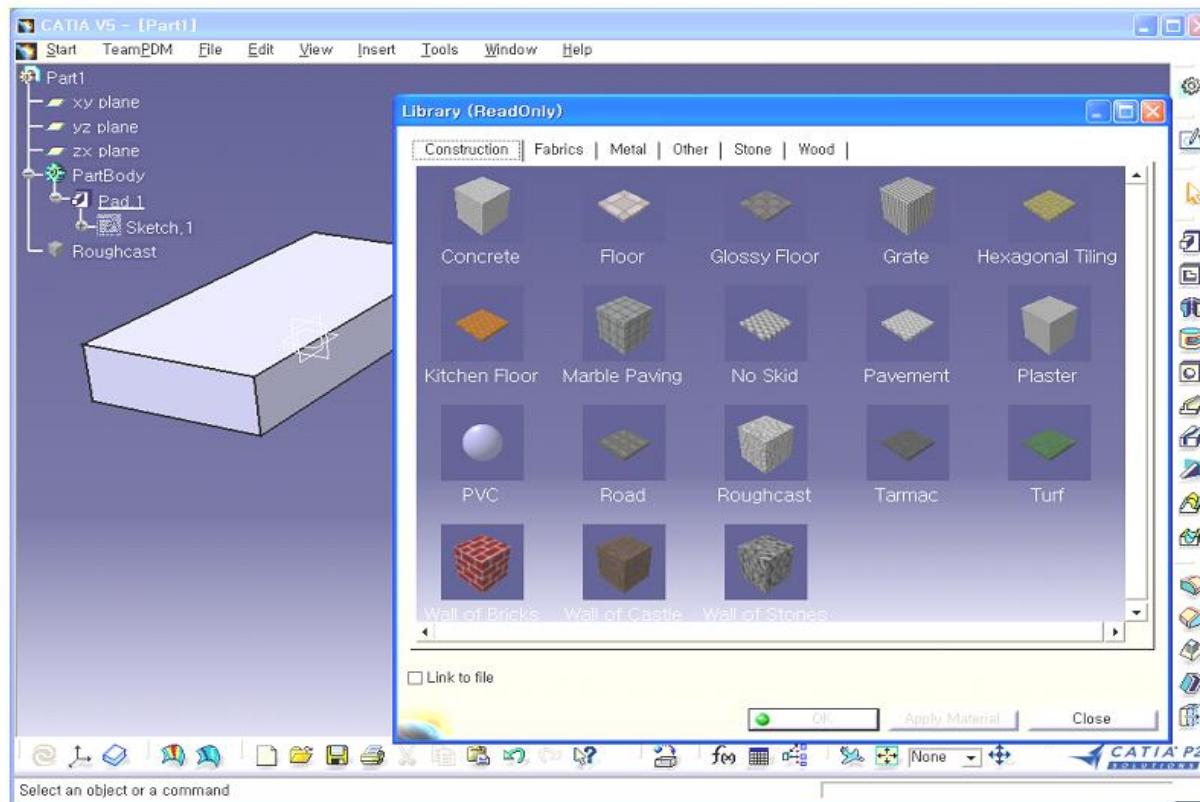
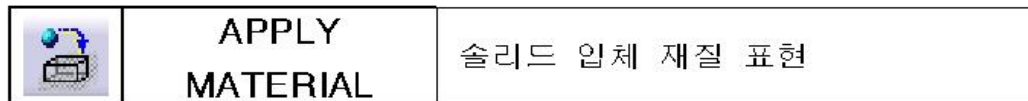


11-2. Constraints



V. PART DESIGN

12. Apply Material

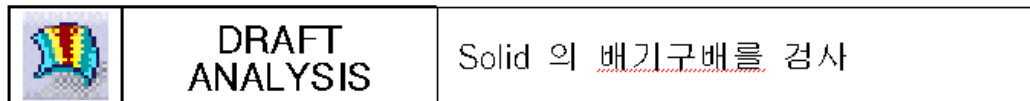


V. PART DESIGN

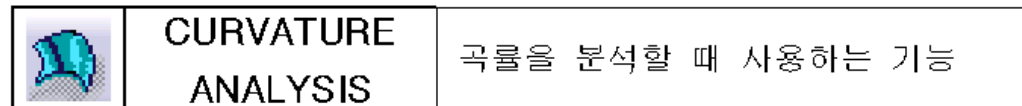
12. Apply Material

Draft 로 생성한 솔리드의 Draft를 분석

12-1. Draft Analysis

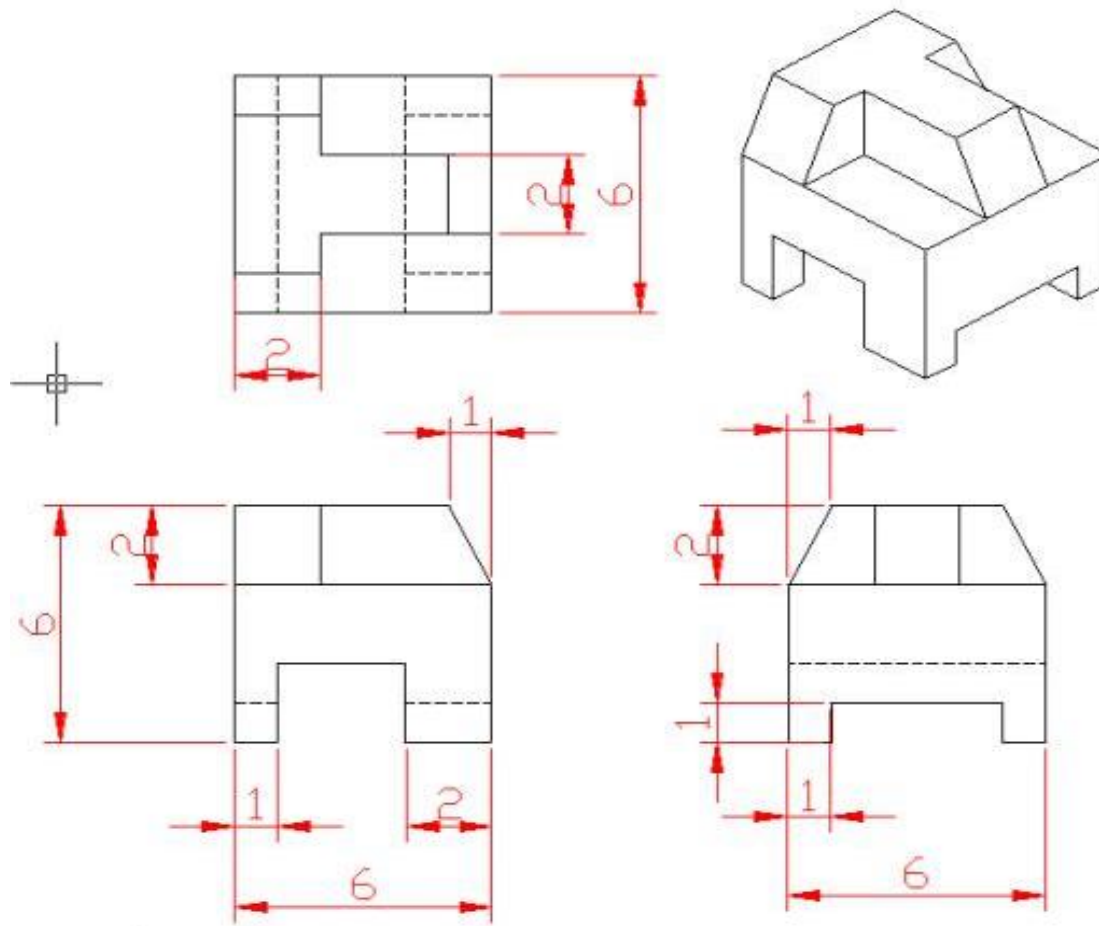


12-2. Curvature Analysis



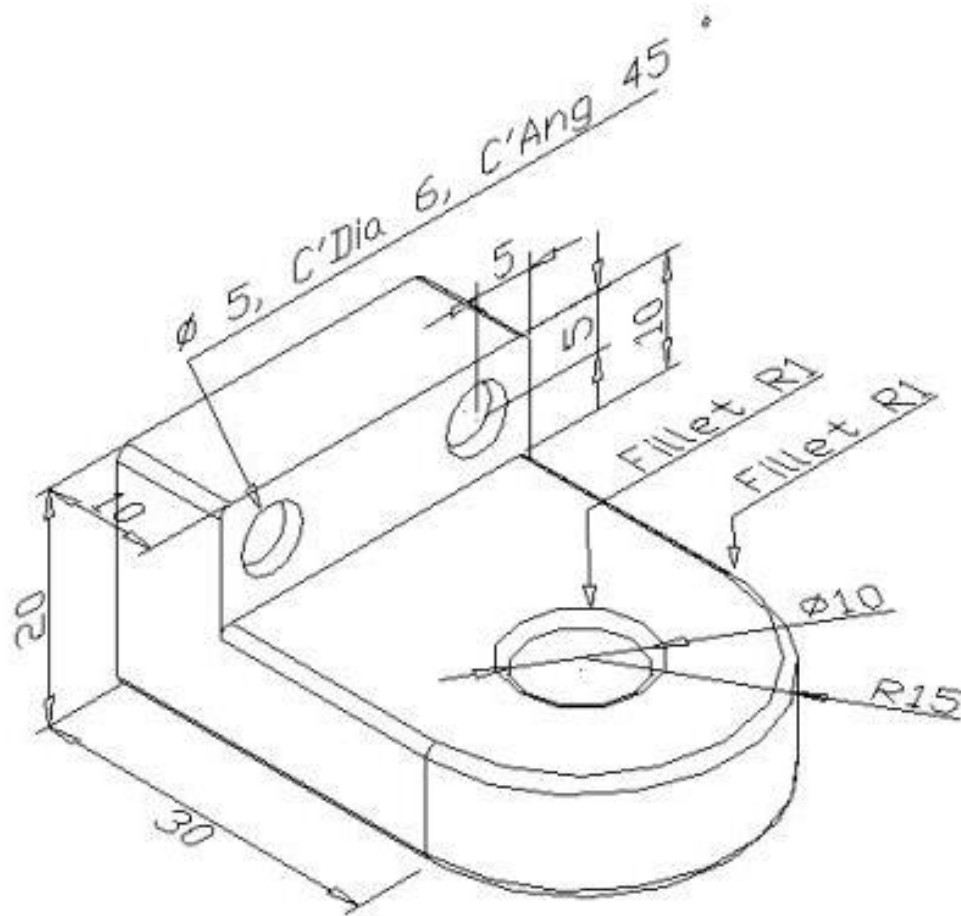
V. PART DESIGN

13. 예 제도면 (1)



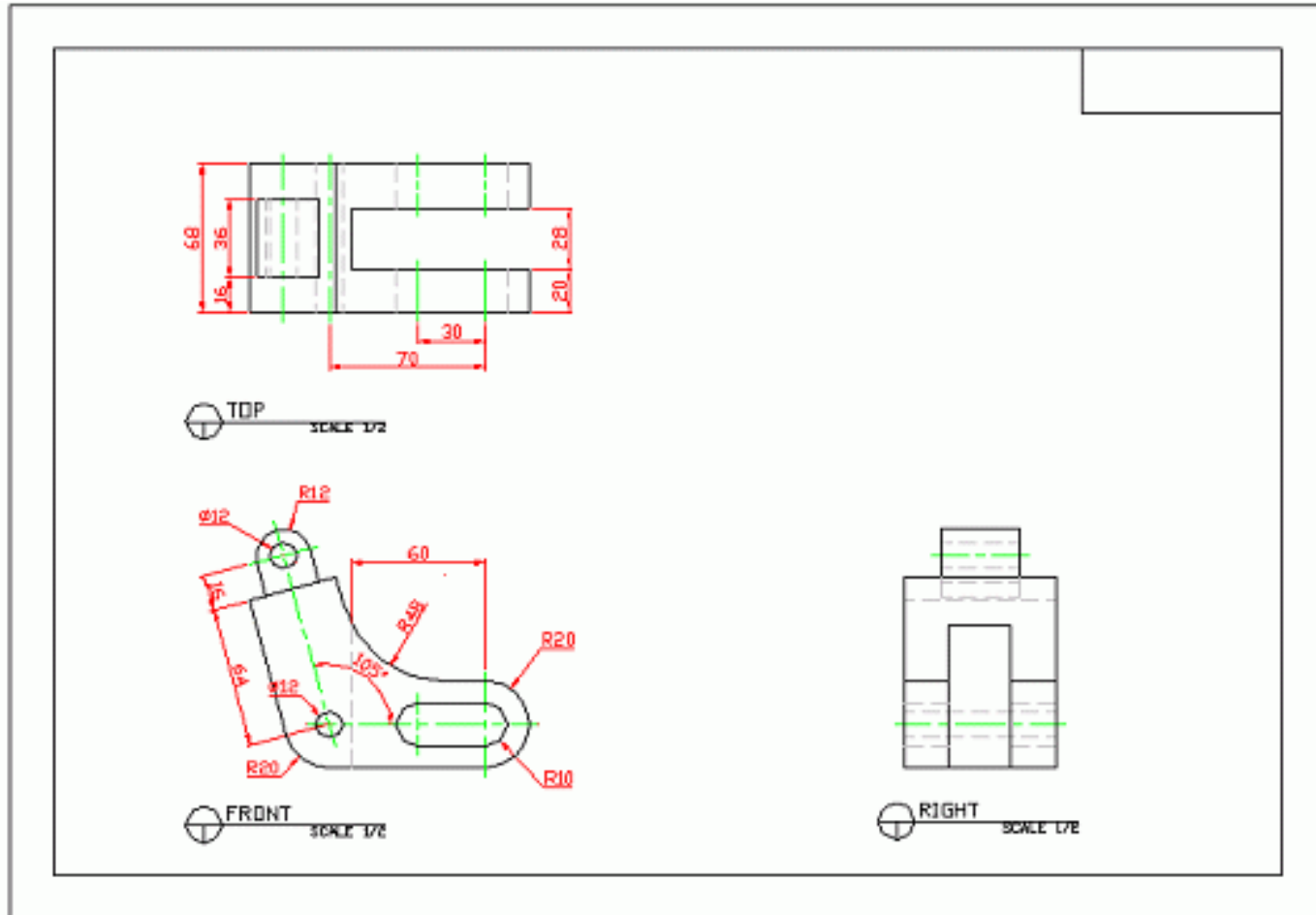
V. PART DESIGN

13. 예 제 도 면 (2)



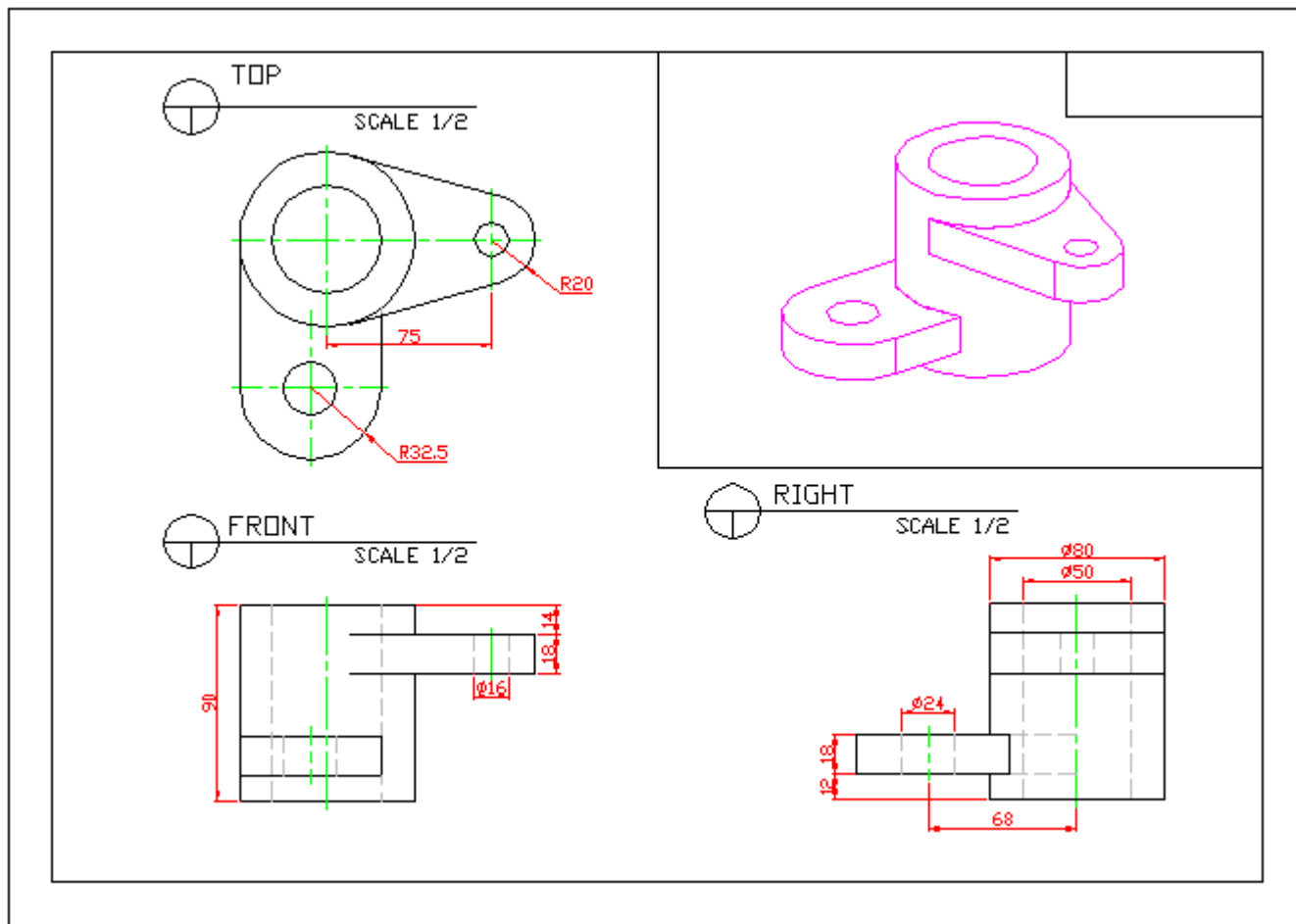
V. PART DESIGN

13. 예 제도면 (3)



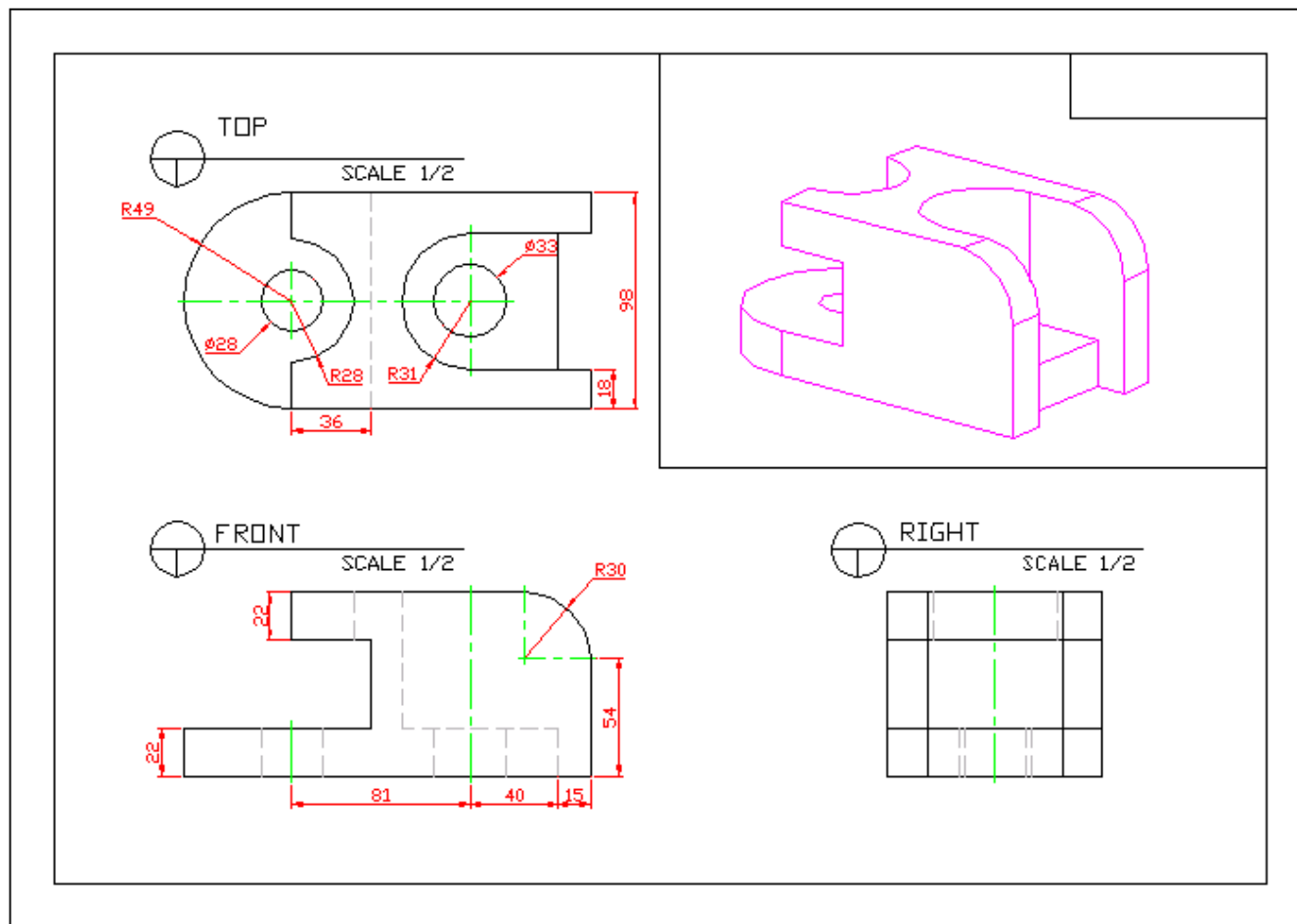
V. PART DESIGN

13. 예 제도면 (4)



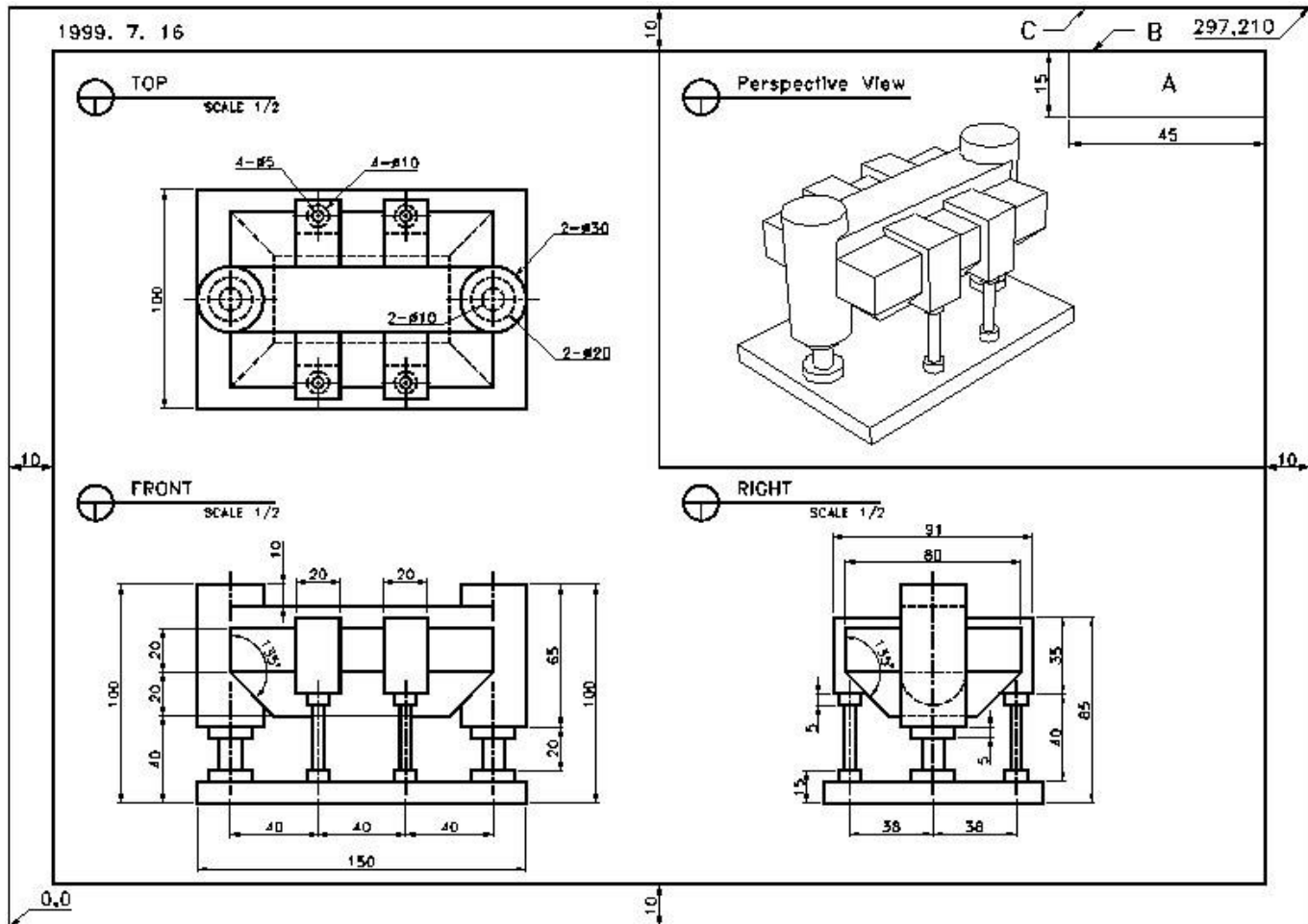
V. PART DESIGN

13. 예 제도면 (5)



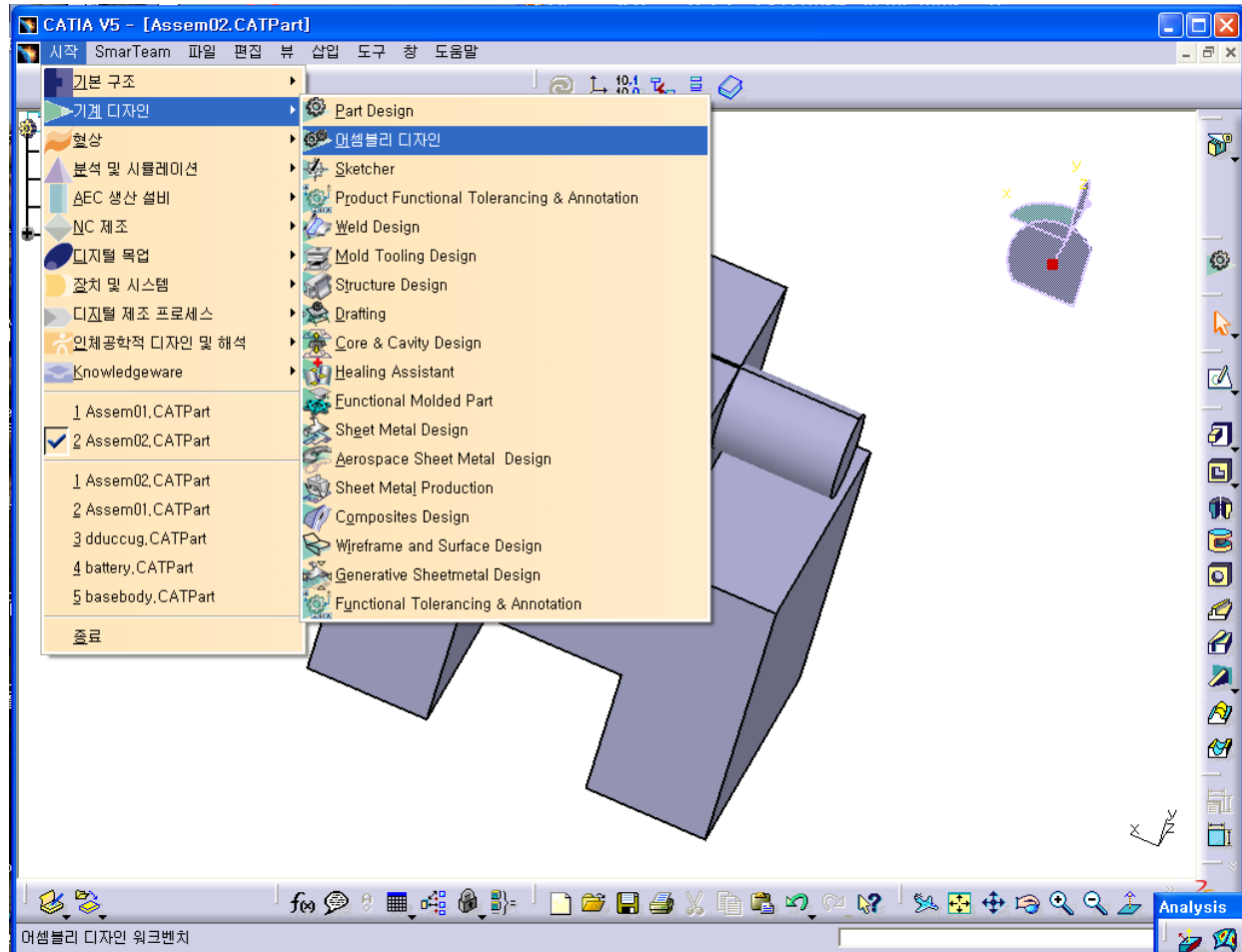
V. PART DESIGN

13. 예 제도면 (7)



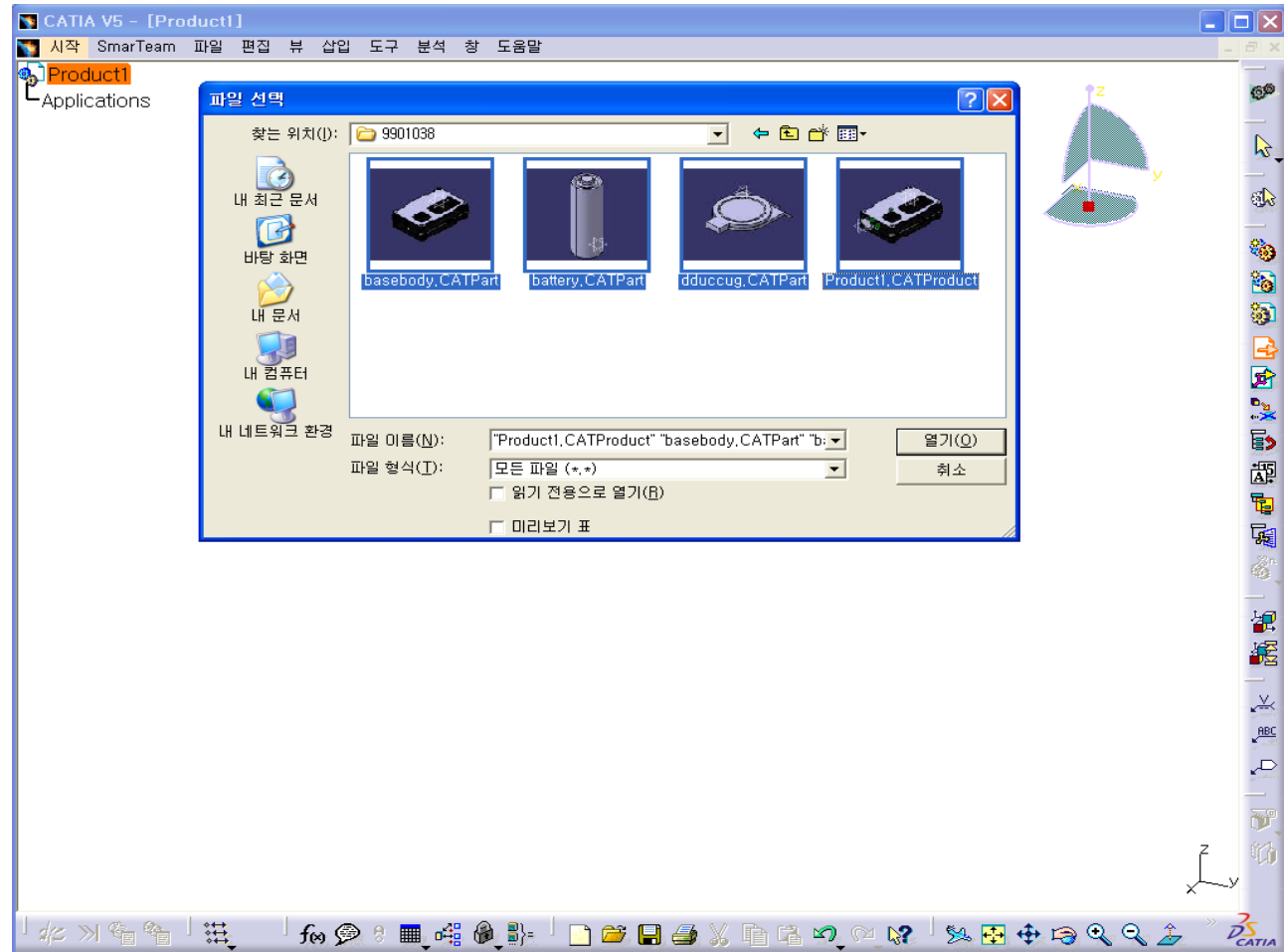
VI. ASSEMBLY DESIGN

1. Introduction (1)



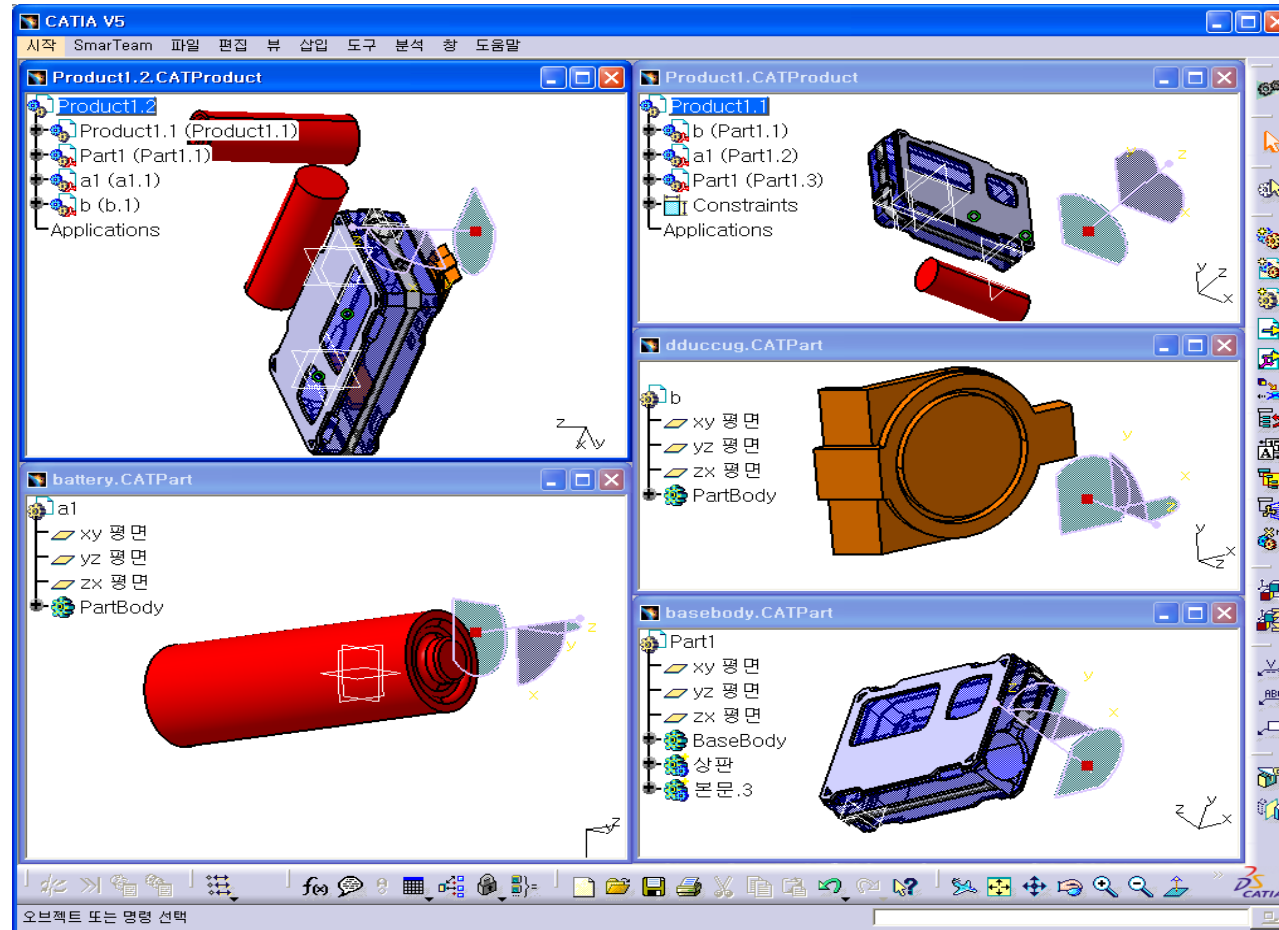
VI. ASSEMBLY DESIGN

1. Introduction (2)



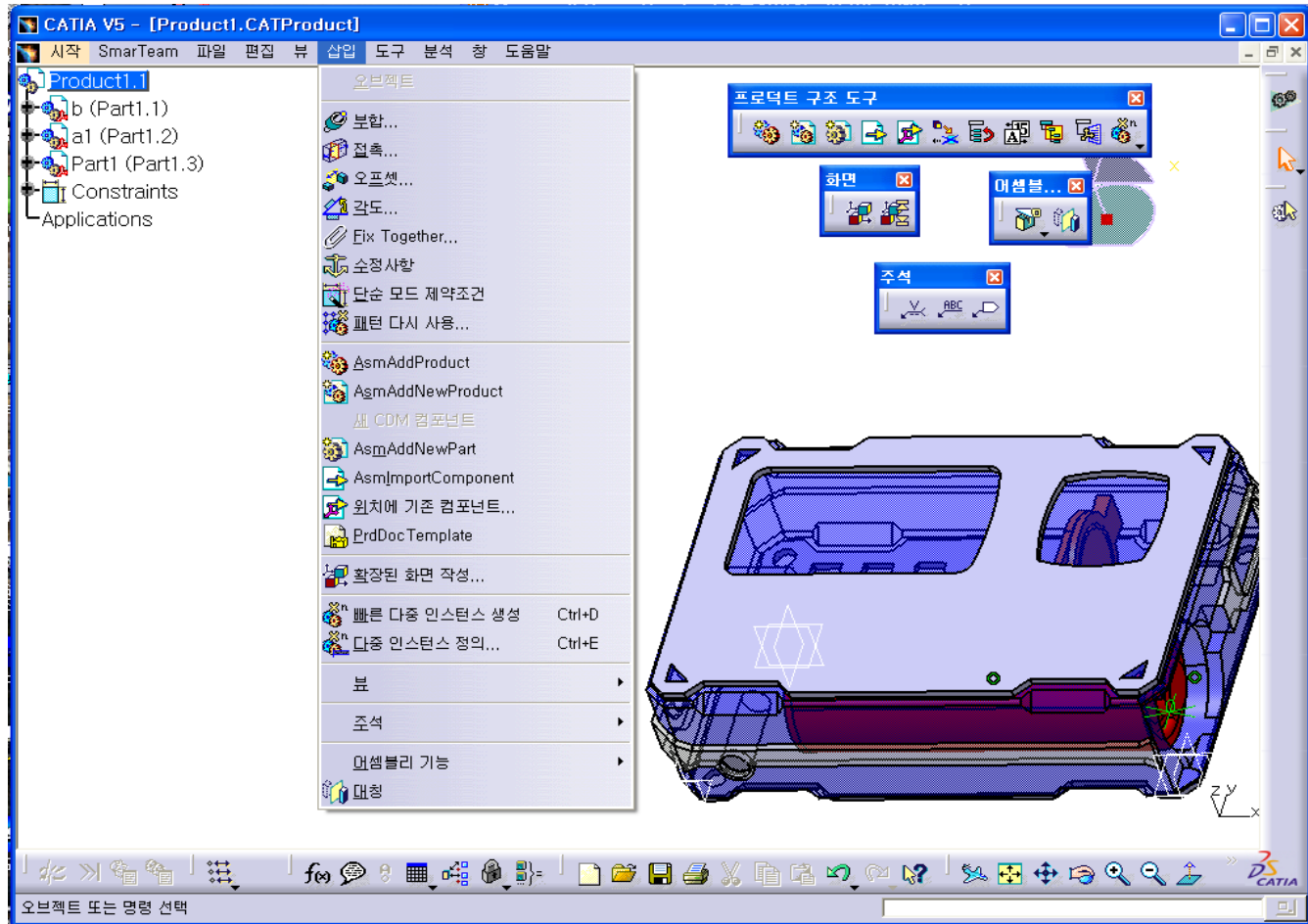
VI. ASSEMBLY DESIGN

1. Introduction (3)



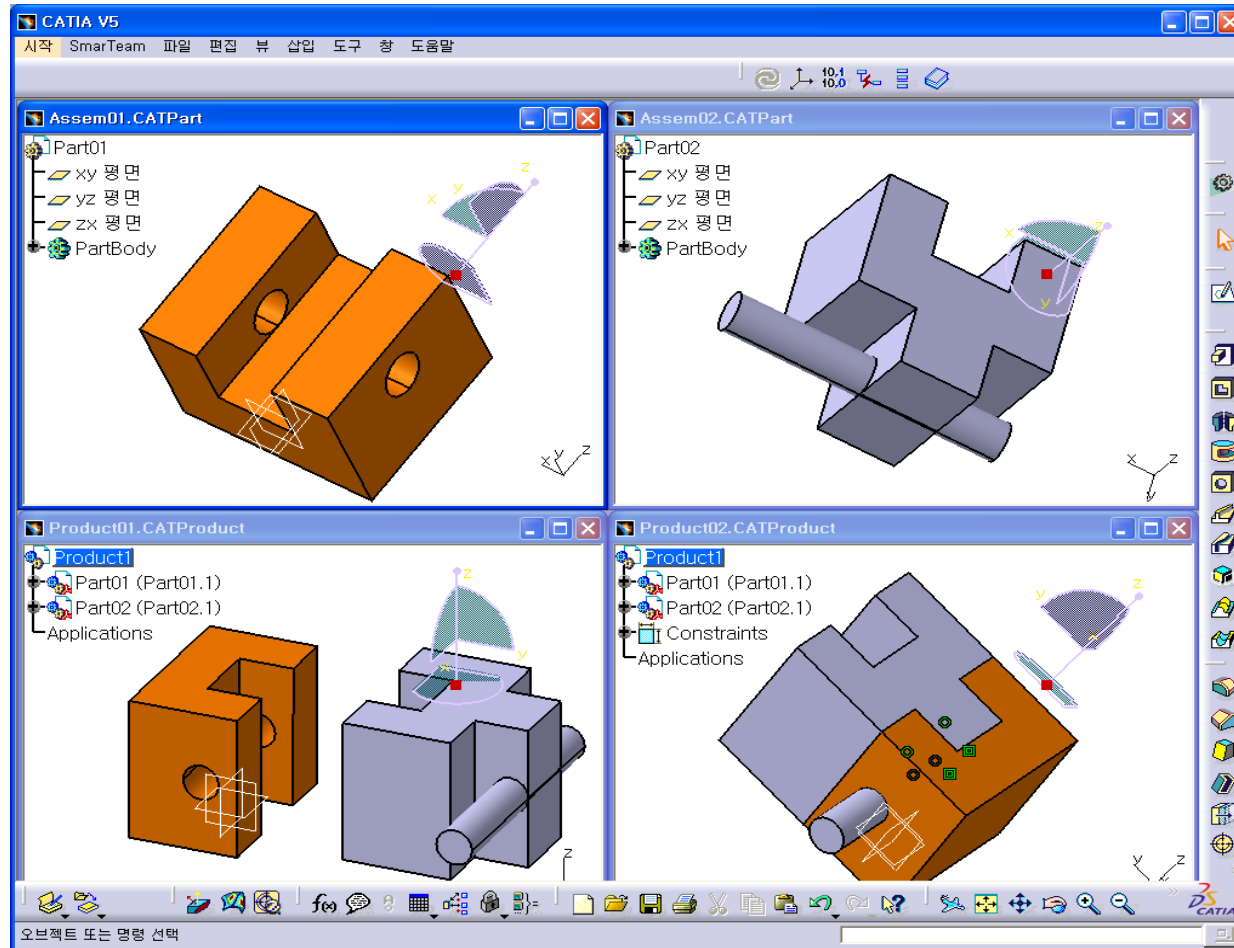
VI. ASSEMBLY DESIGN

1. Introduction (4)



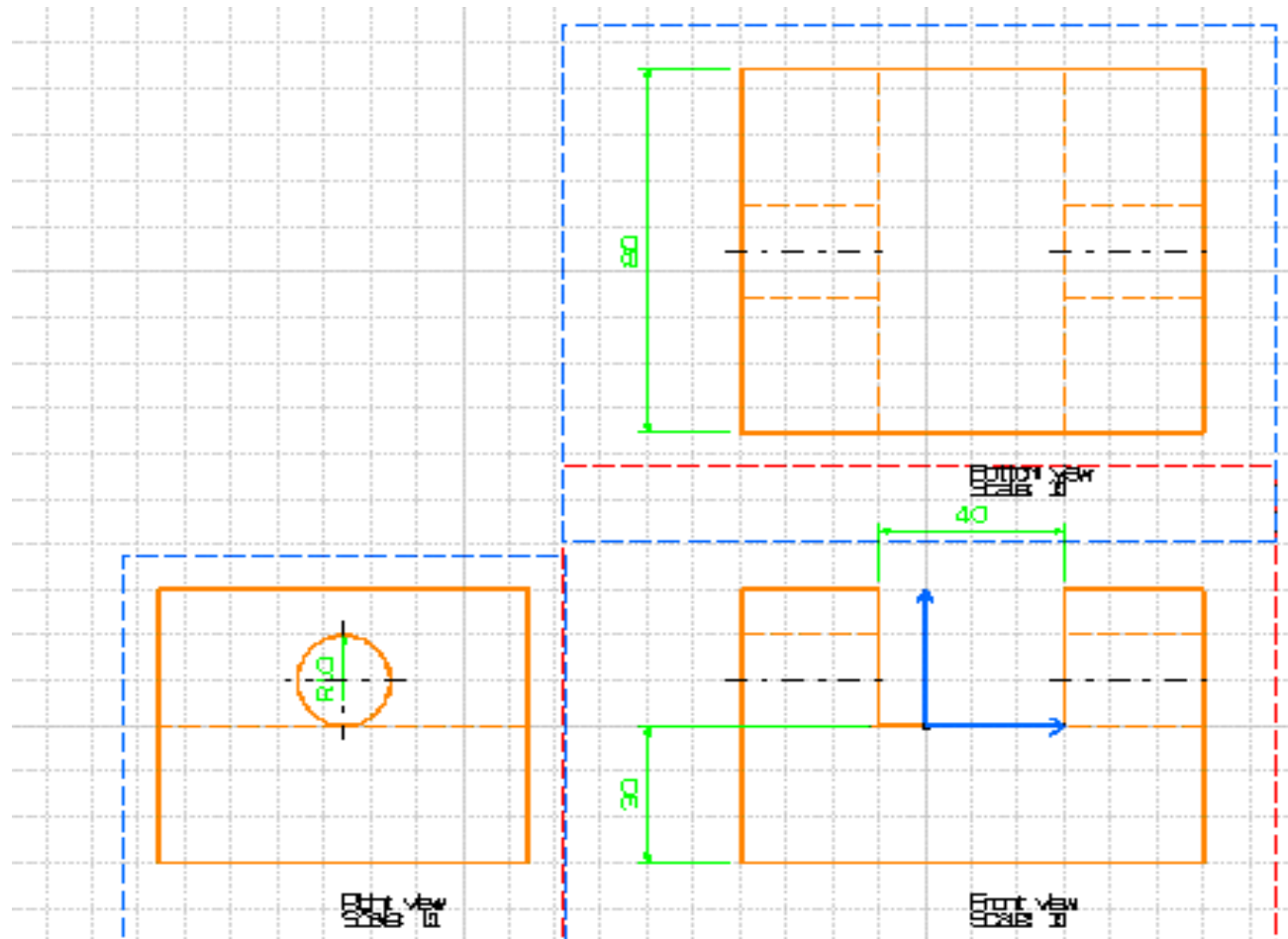
VI. ASSEMBLY DESIGN

2. 실습 과제



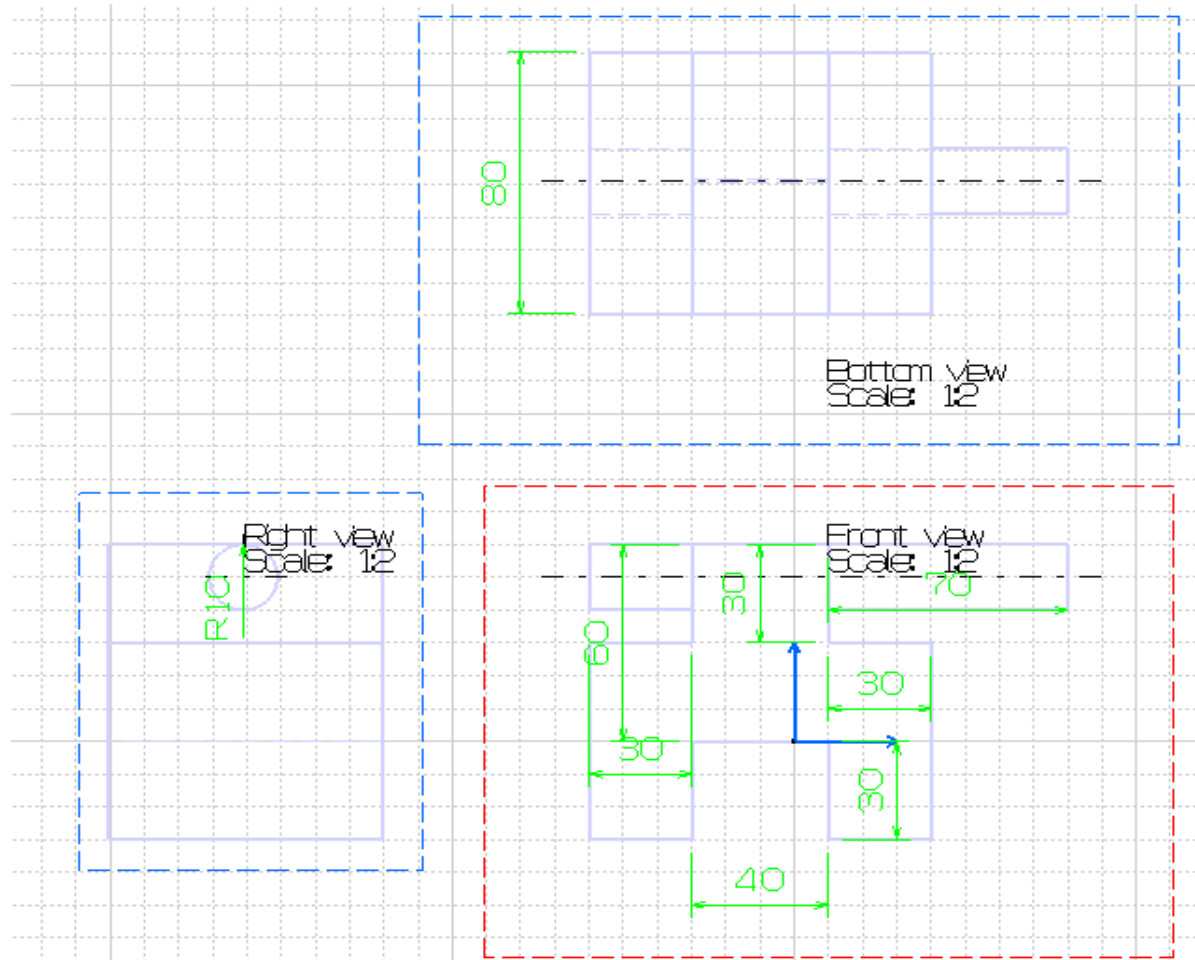
VI. ASSEMBLY DESIGN

3. 모델링 (1)



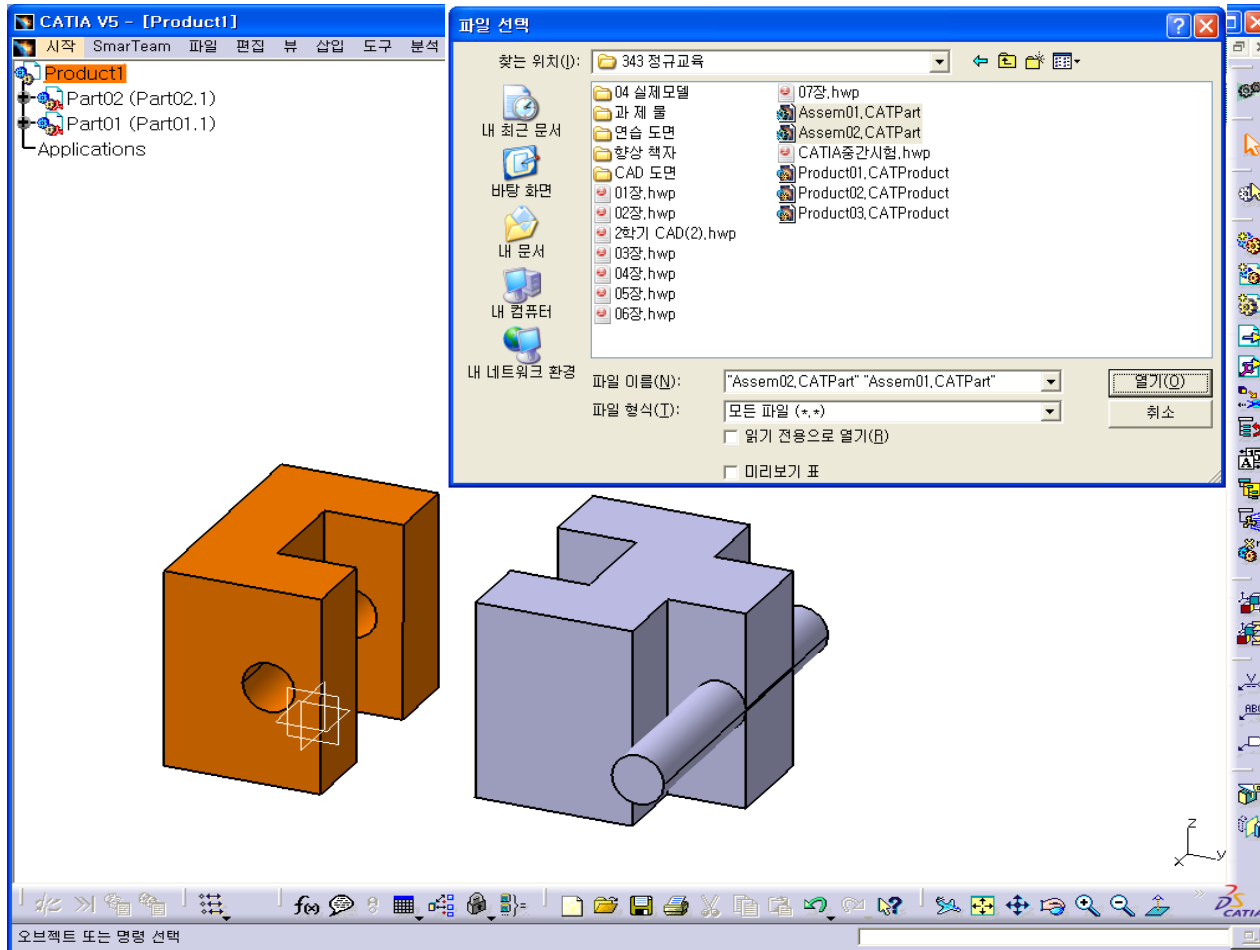
VI. ASSEMBLY DESIGN

3. 모델링 (2)



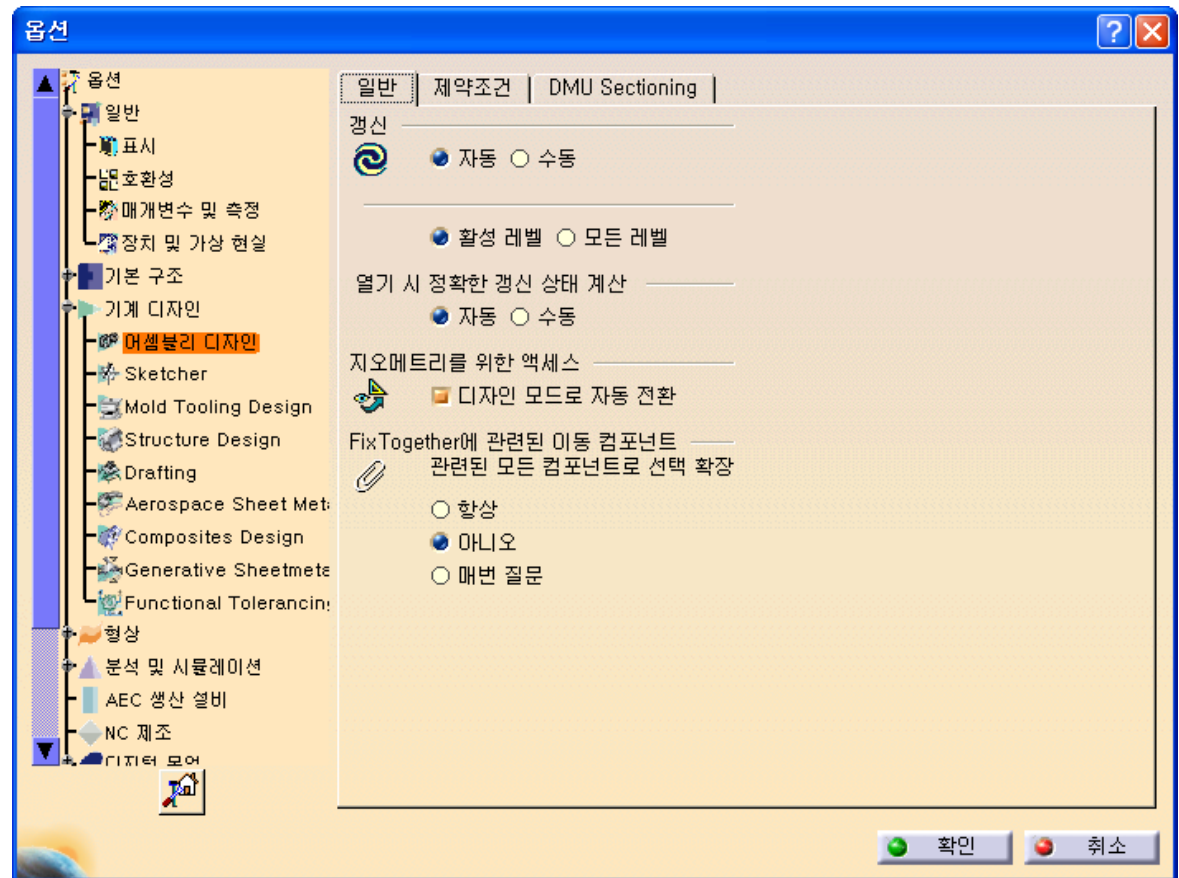
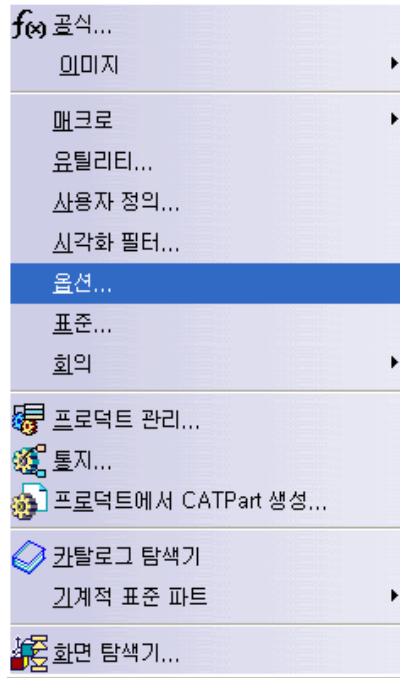
VI. ASSEMBLY DESIGN

4. 어셈블리 (1)



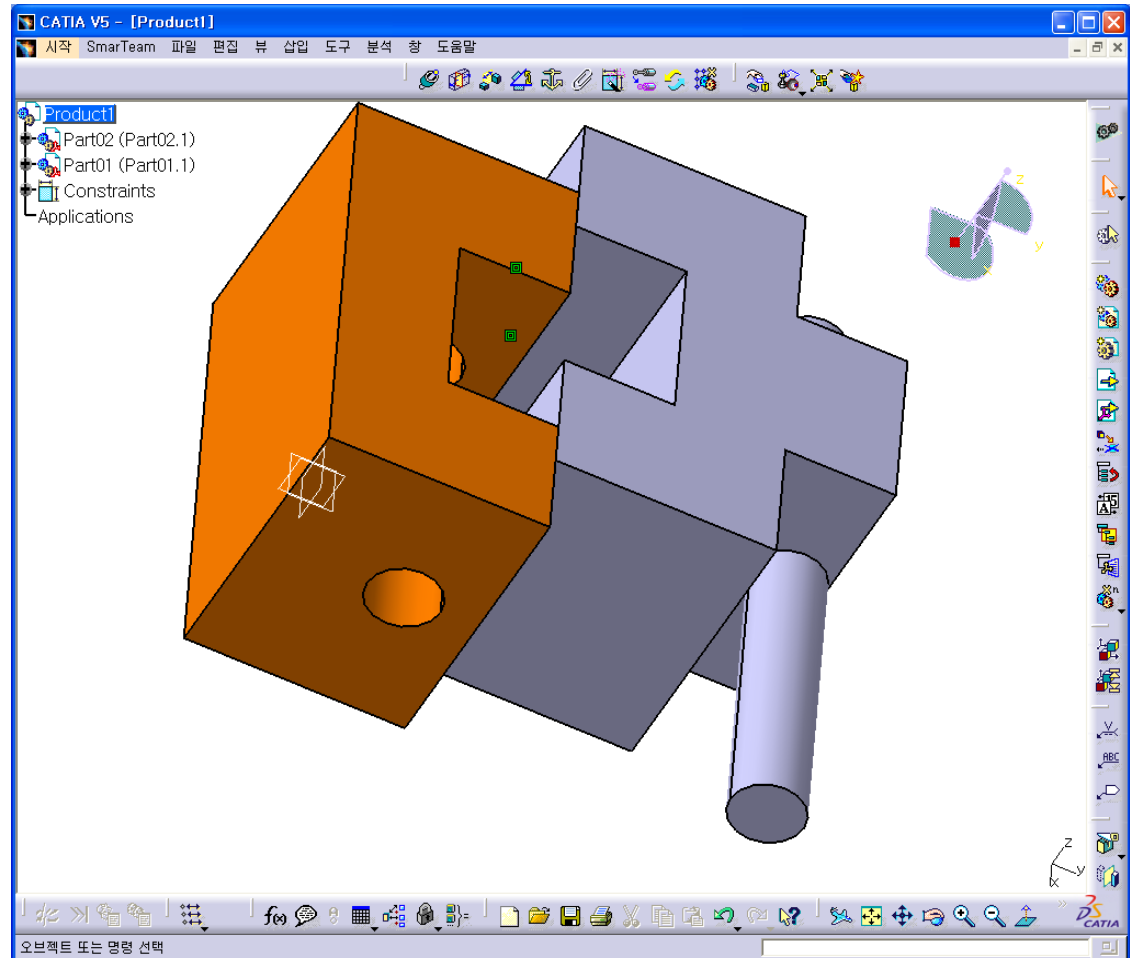
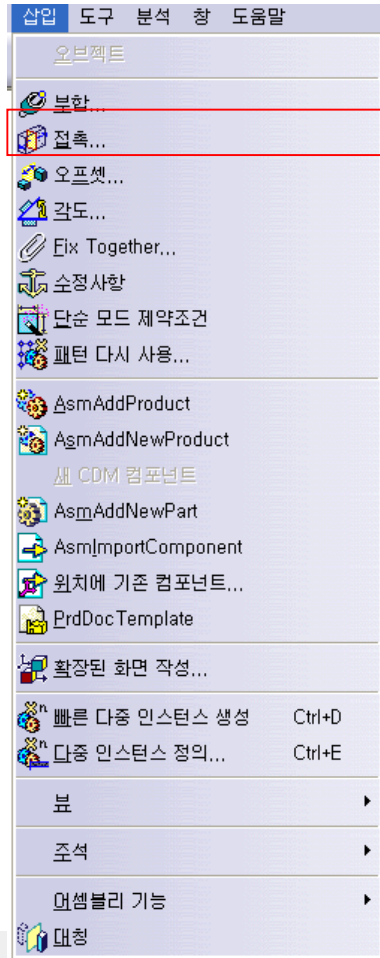
VI. ASSEMBLY DESIGN

4. 어셈블리 (2)



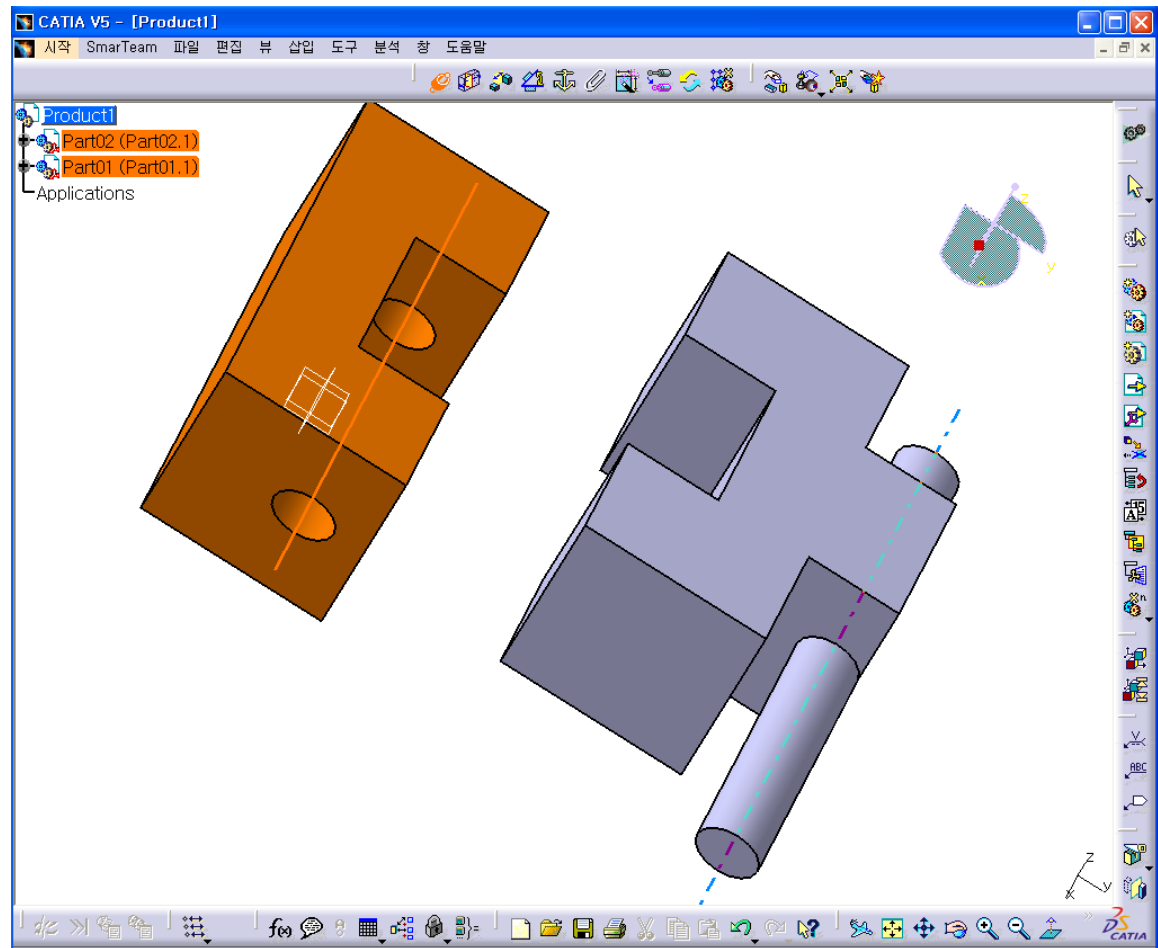
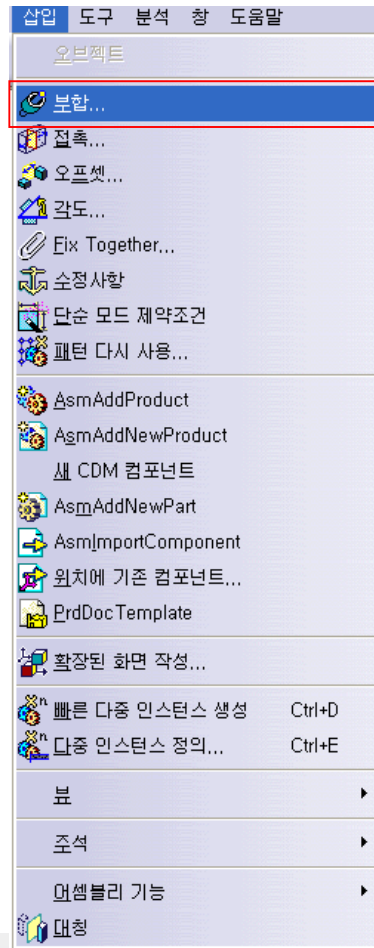
VI. ASSEMBLY DESIGN

4. 어셈블리 (3)



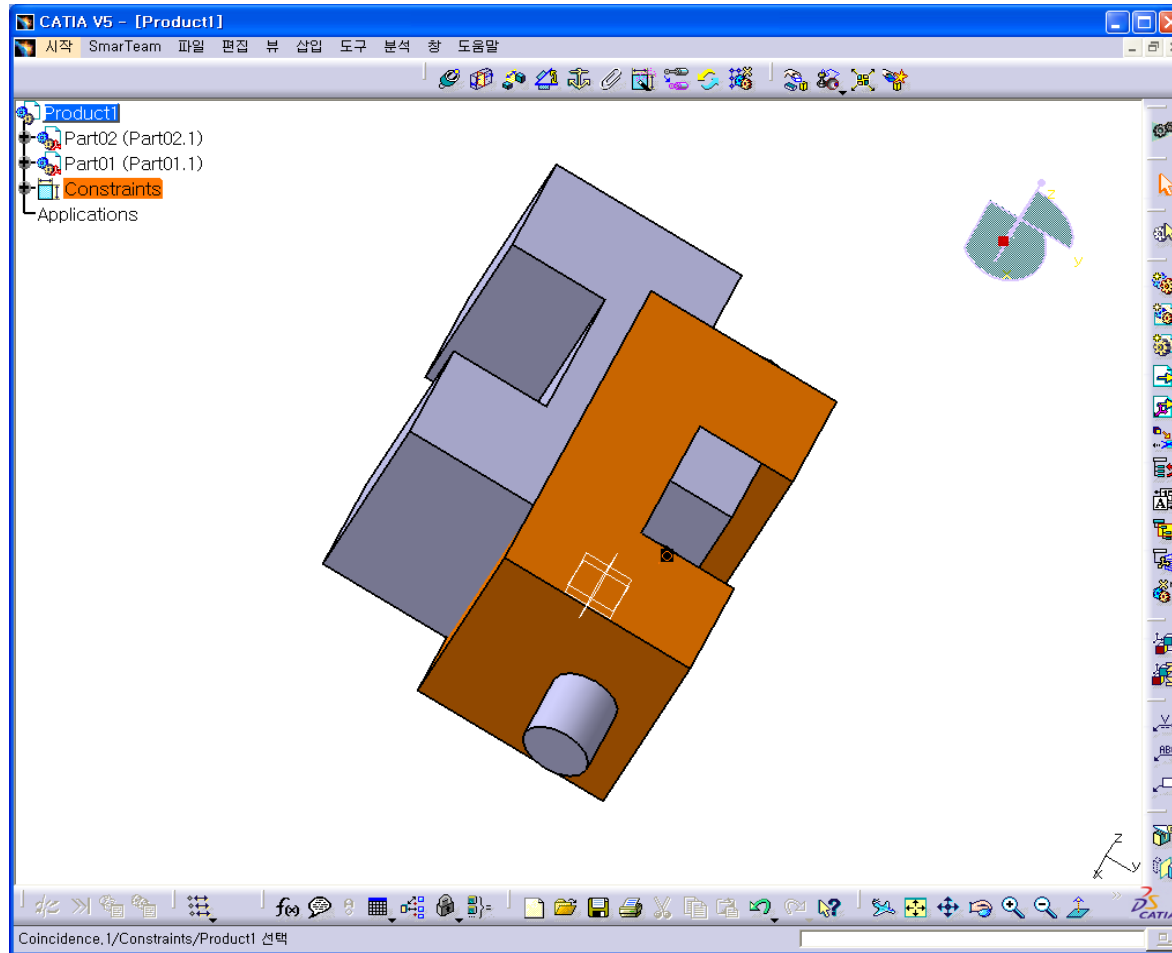
VI. ASSEMBLY DESIGN

4. 어셈블리 (4)



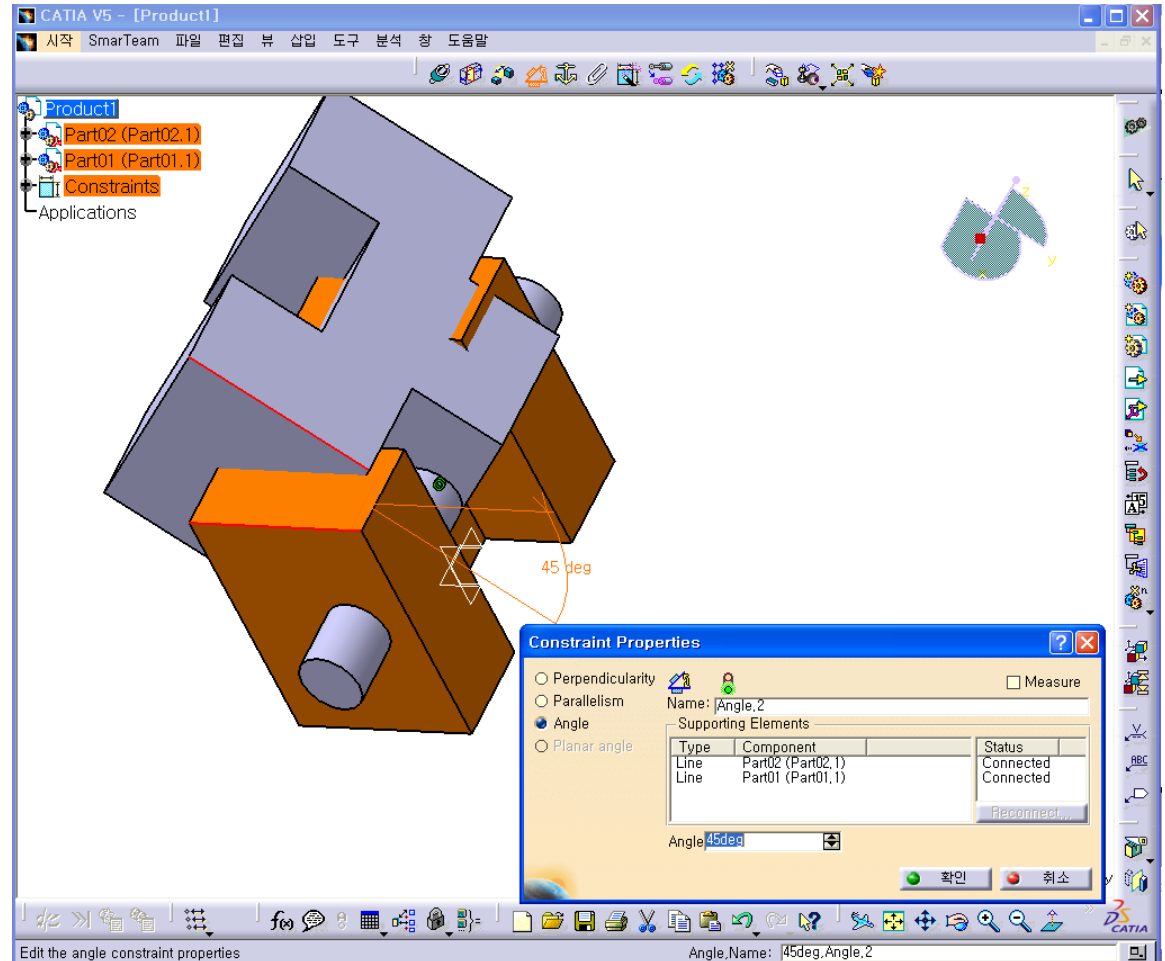
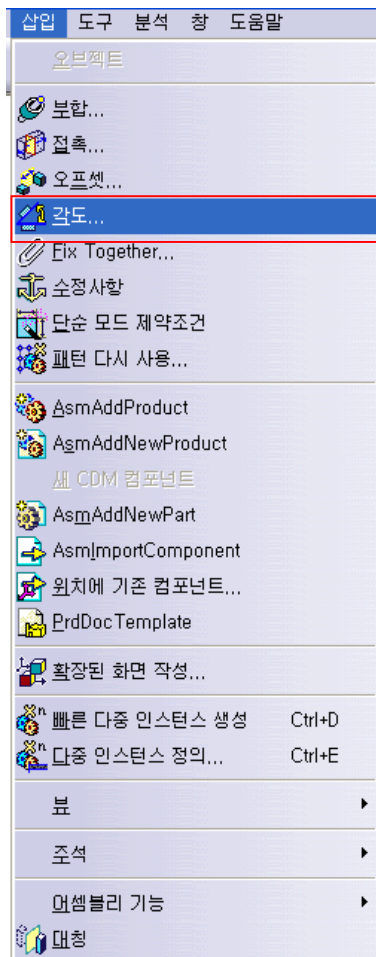
VI. ASSEMBLY DESIGN

4. 어셈블리 (5)



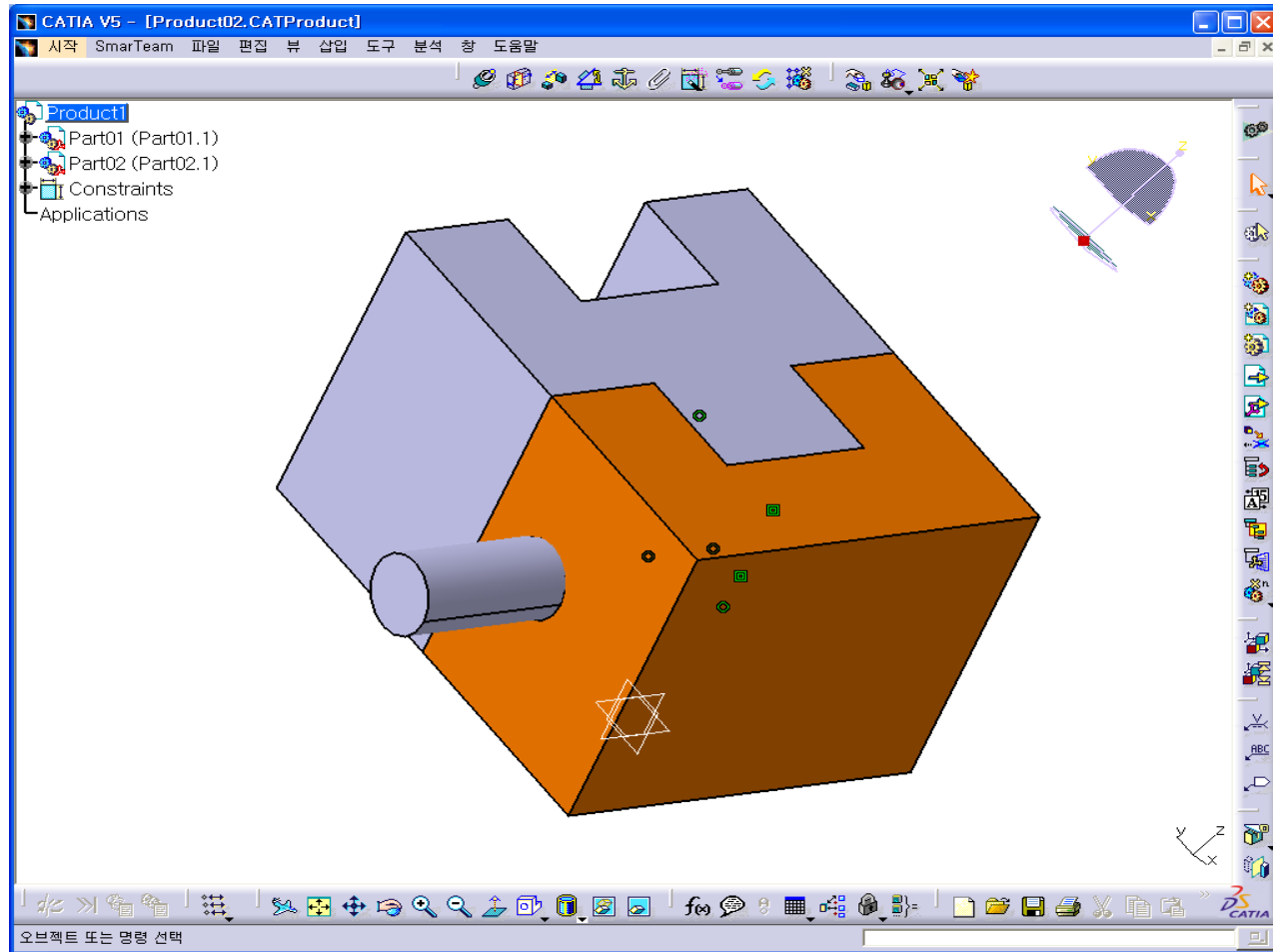
VI. ASSEMBLY DESIGN

4. 어셈블리 (6)



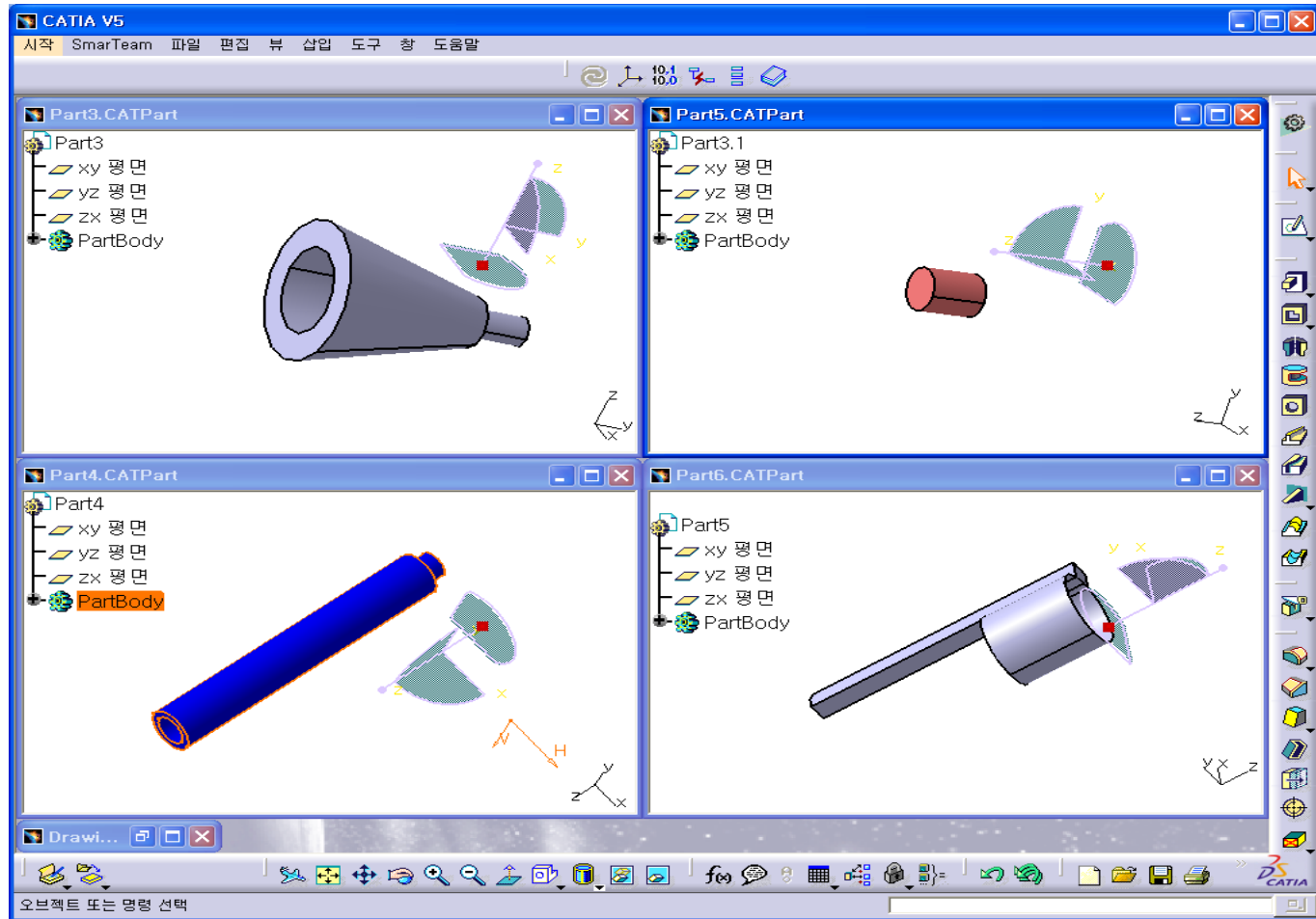
VI. ASSEMBLY DESIGN

4. 어셈블리 (7)



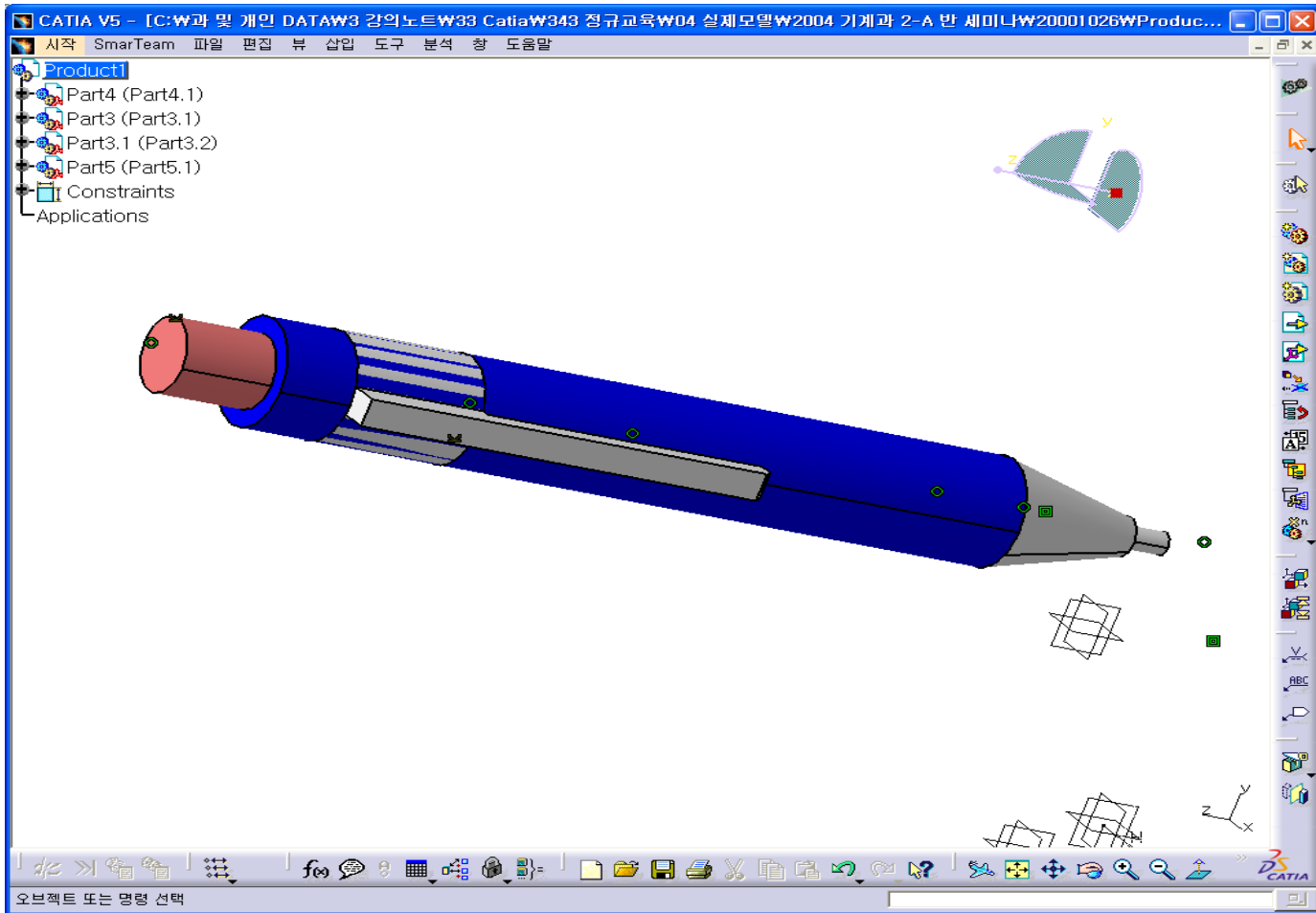
VI. ASSEMBLY DESIGN

5. 어셈블리예제 (1)



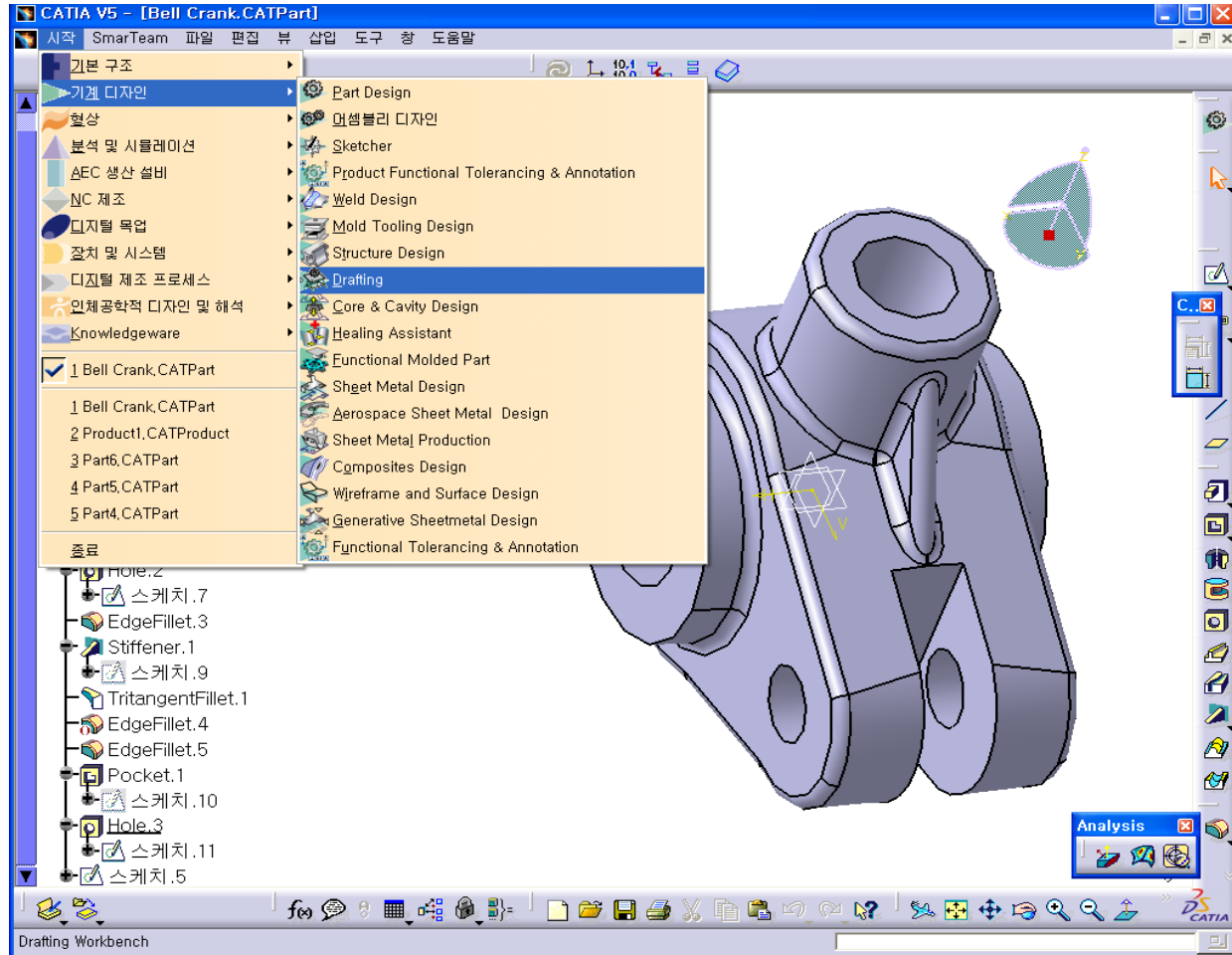
VI. ASSEMBLY DESIGN

5. 어셈블리예제 (2)



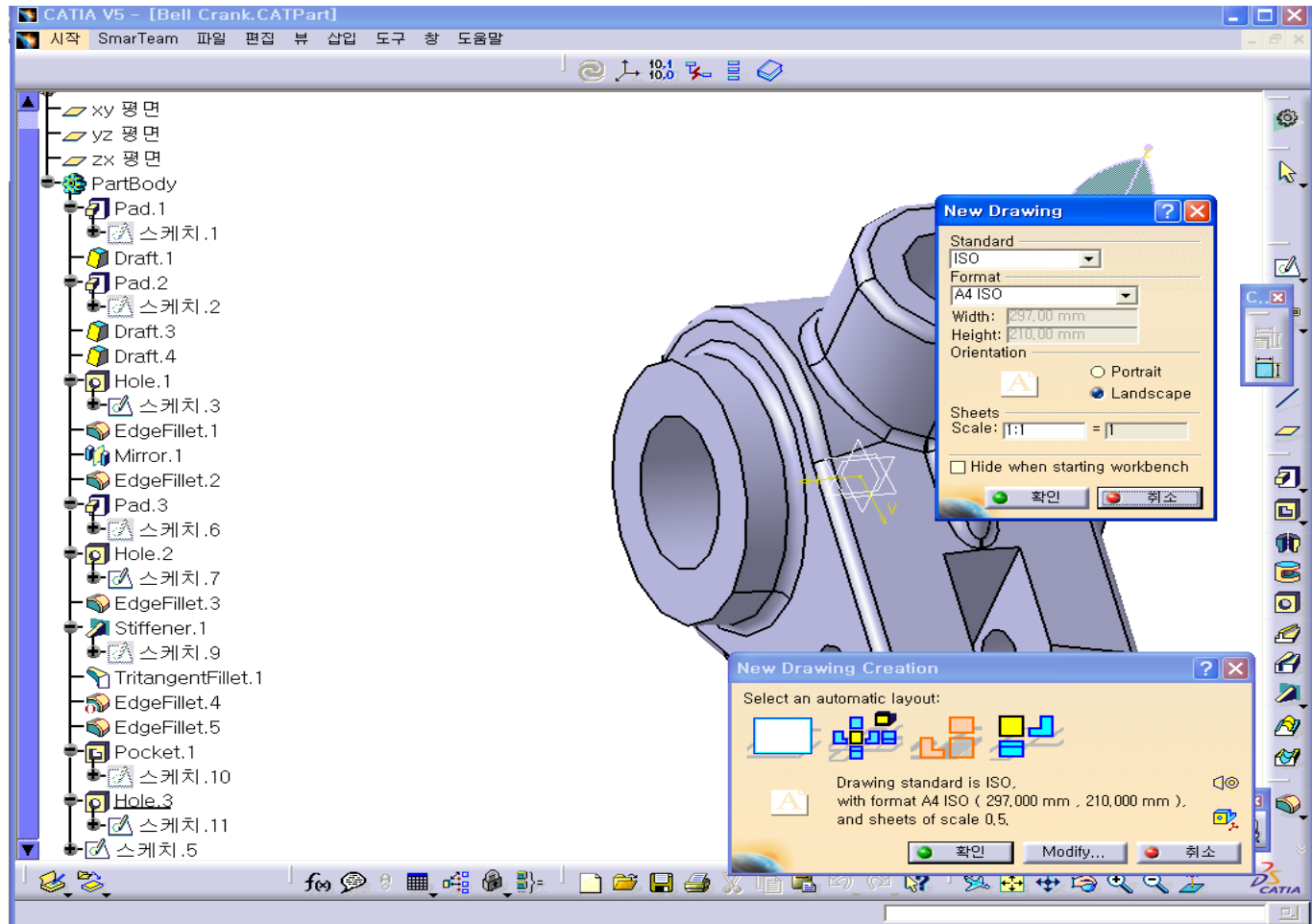
VII. DRAFTING

1. Introduction (1)



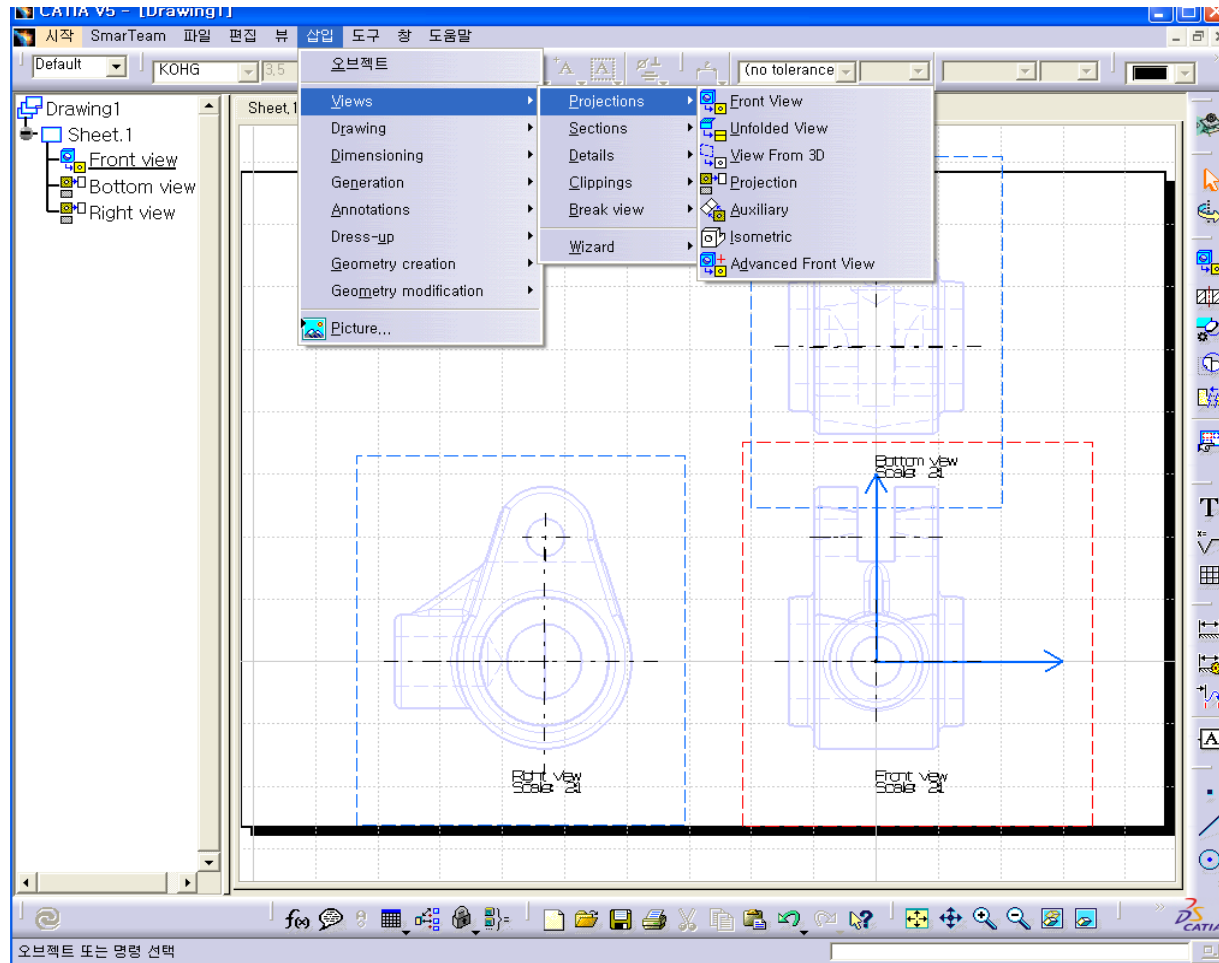
VII. DRAFTING

1. Introduction (2)



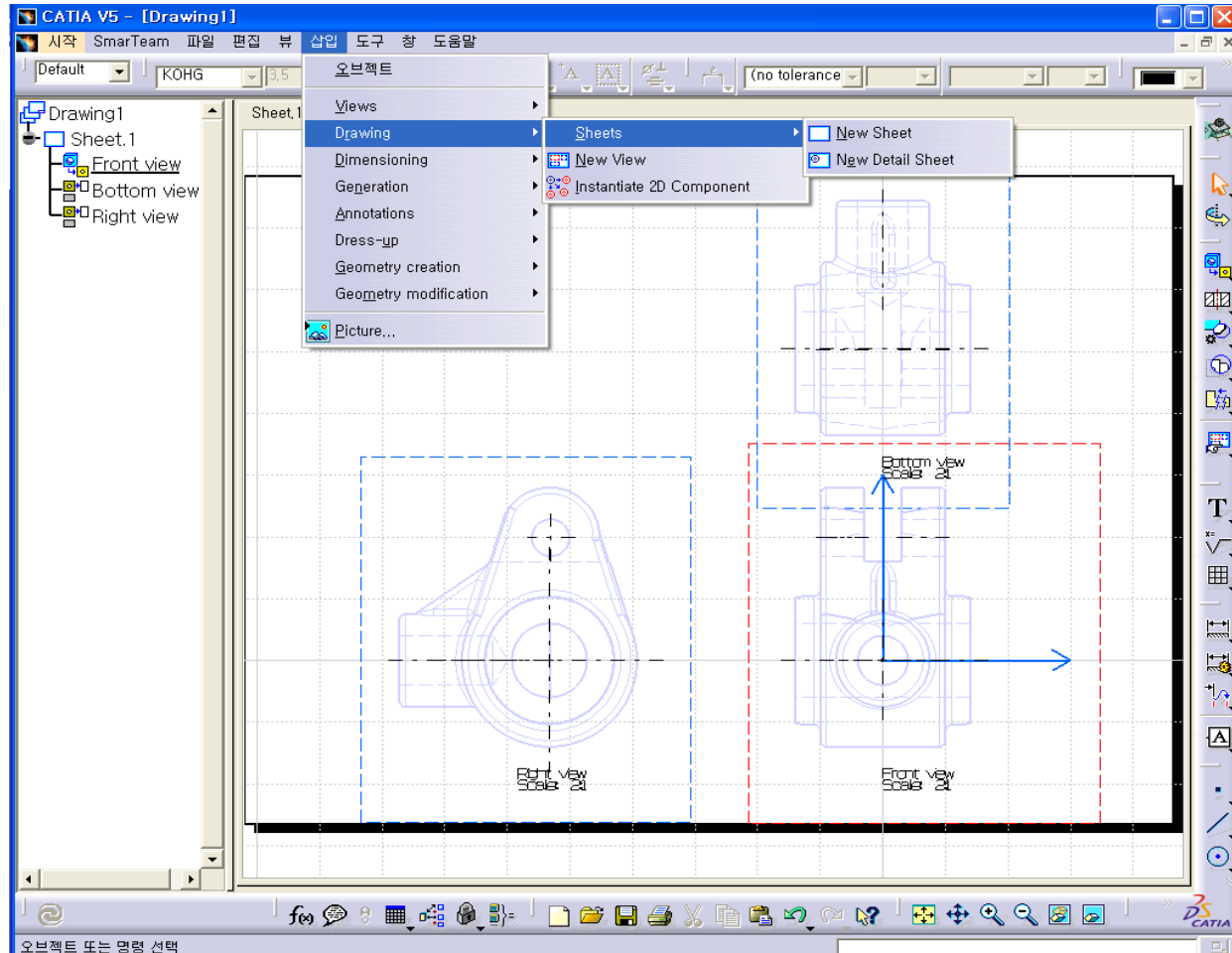
VII. DRAFTING

1. Introduction (3)



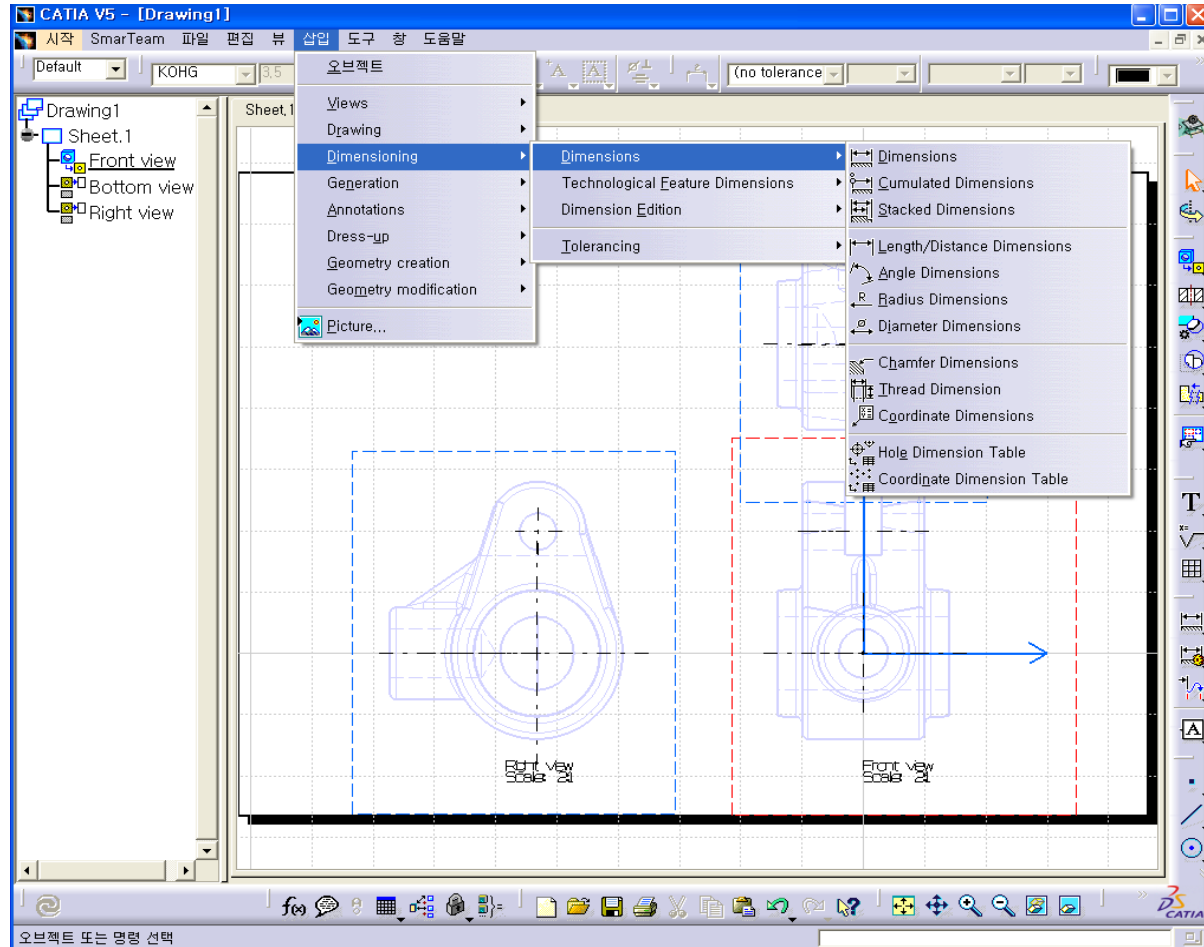
VII. DRAFTING

1. Introduction (4)



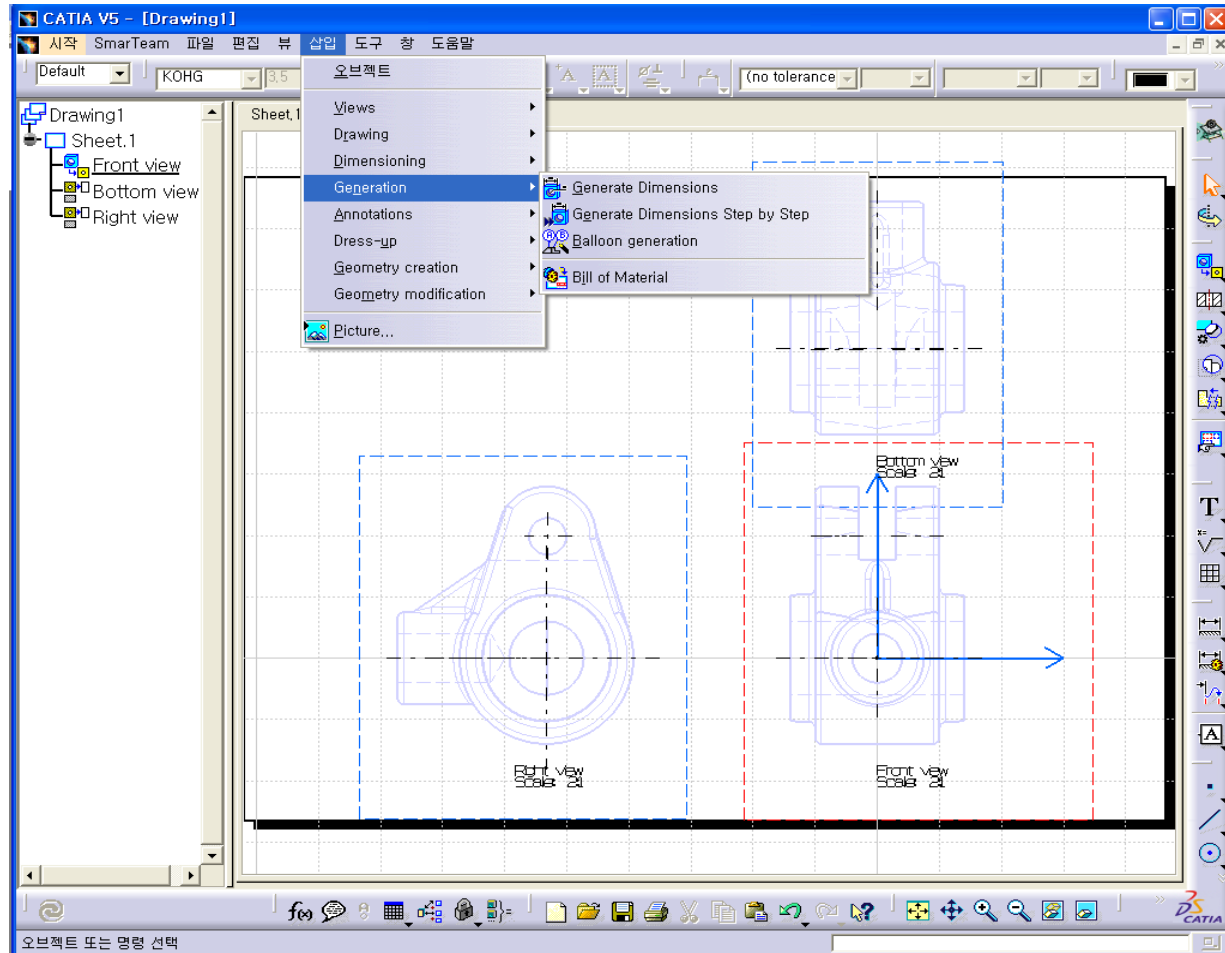
VII. DRAFTING

1. Introduction (5)



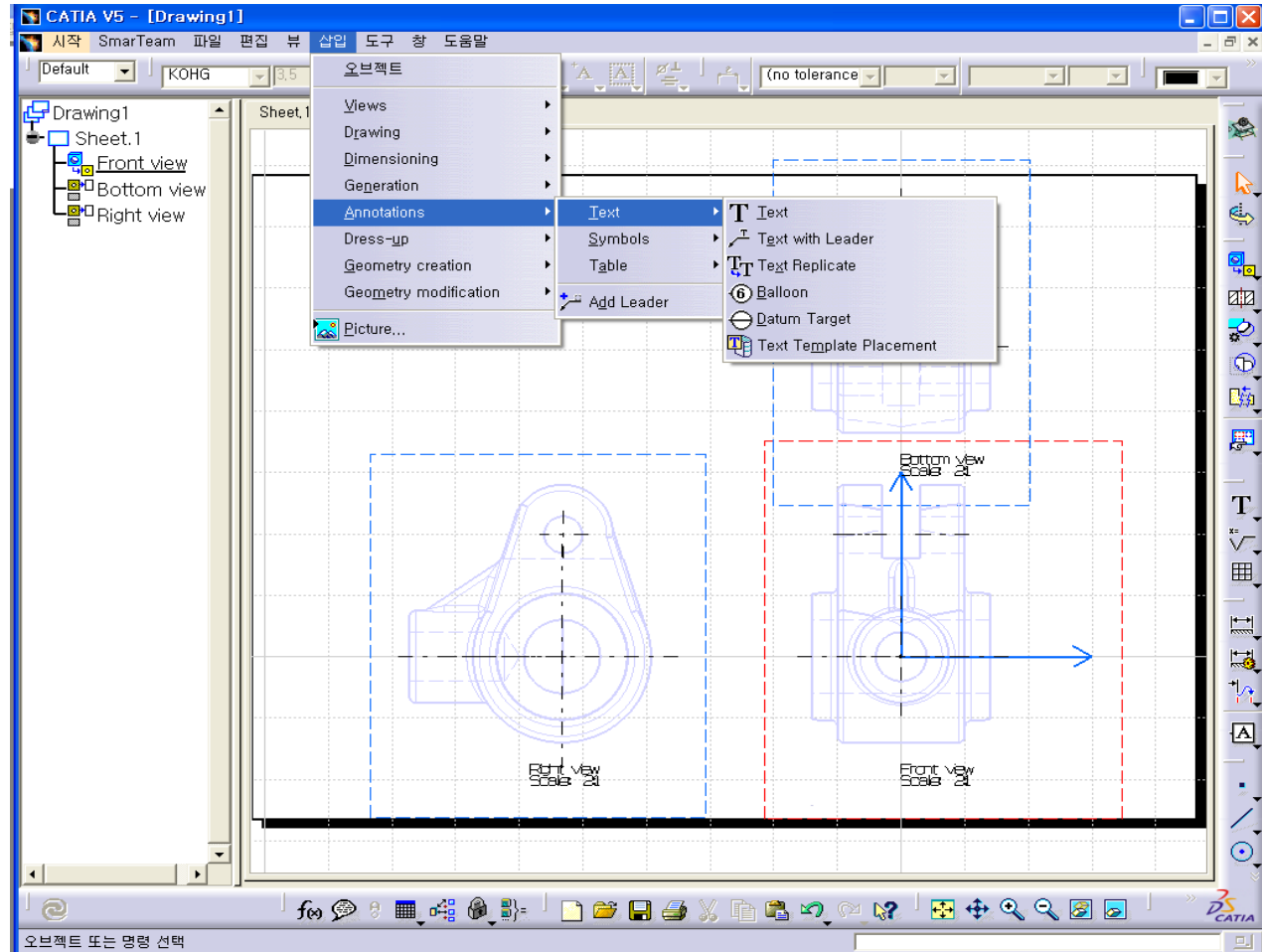
VII. DRAFTING

1. Introduction (6)



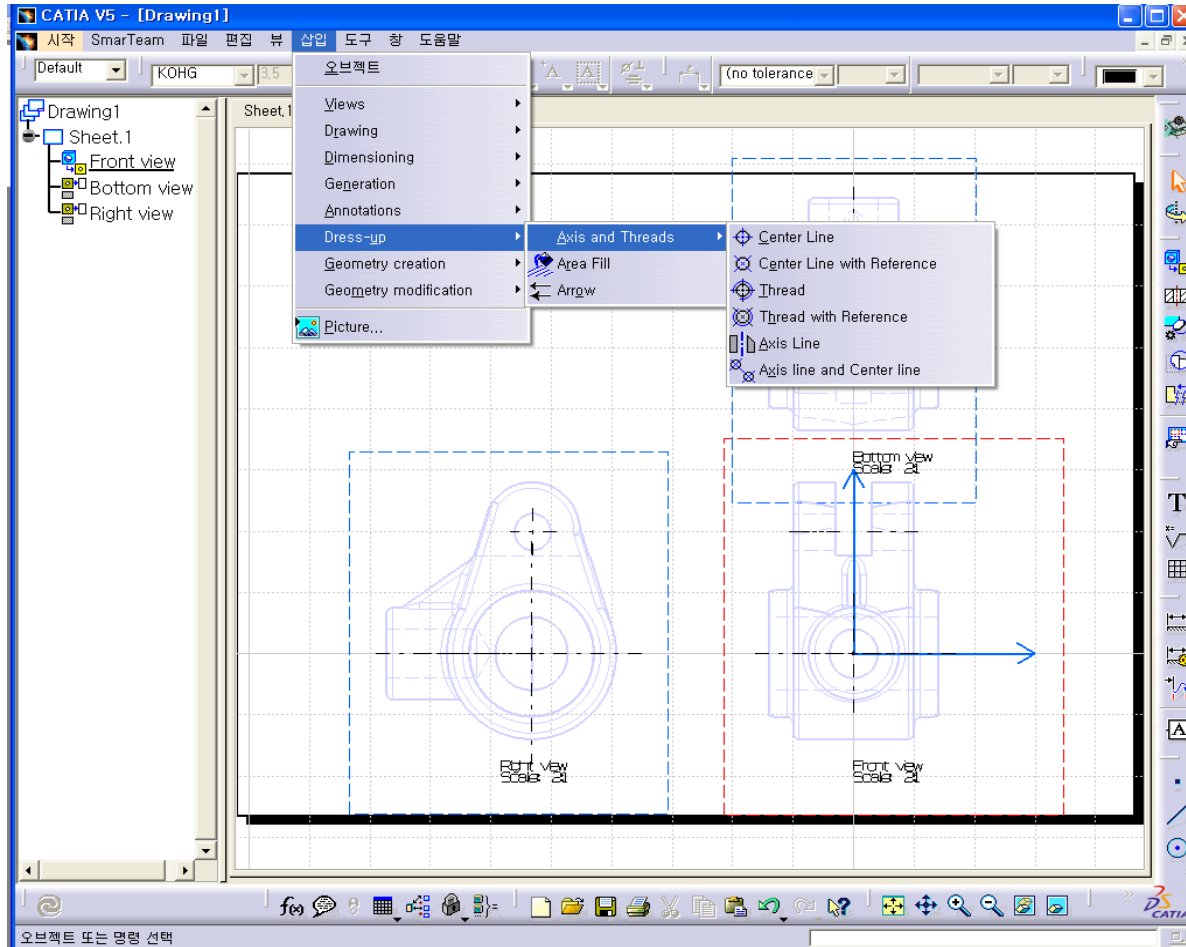
VII. DRAFTING

1. Introduction (7)



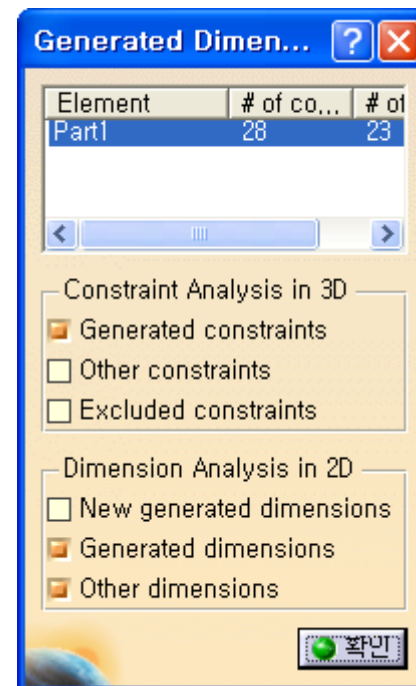
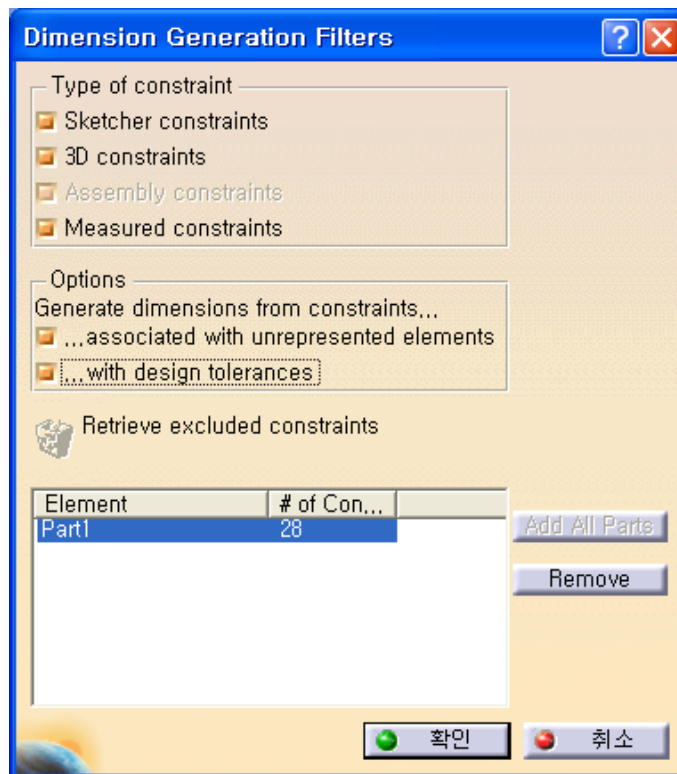
VII. DRAFTING

1. Introduction (8)



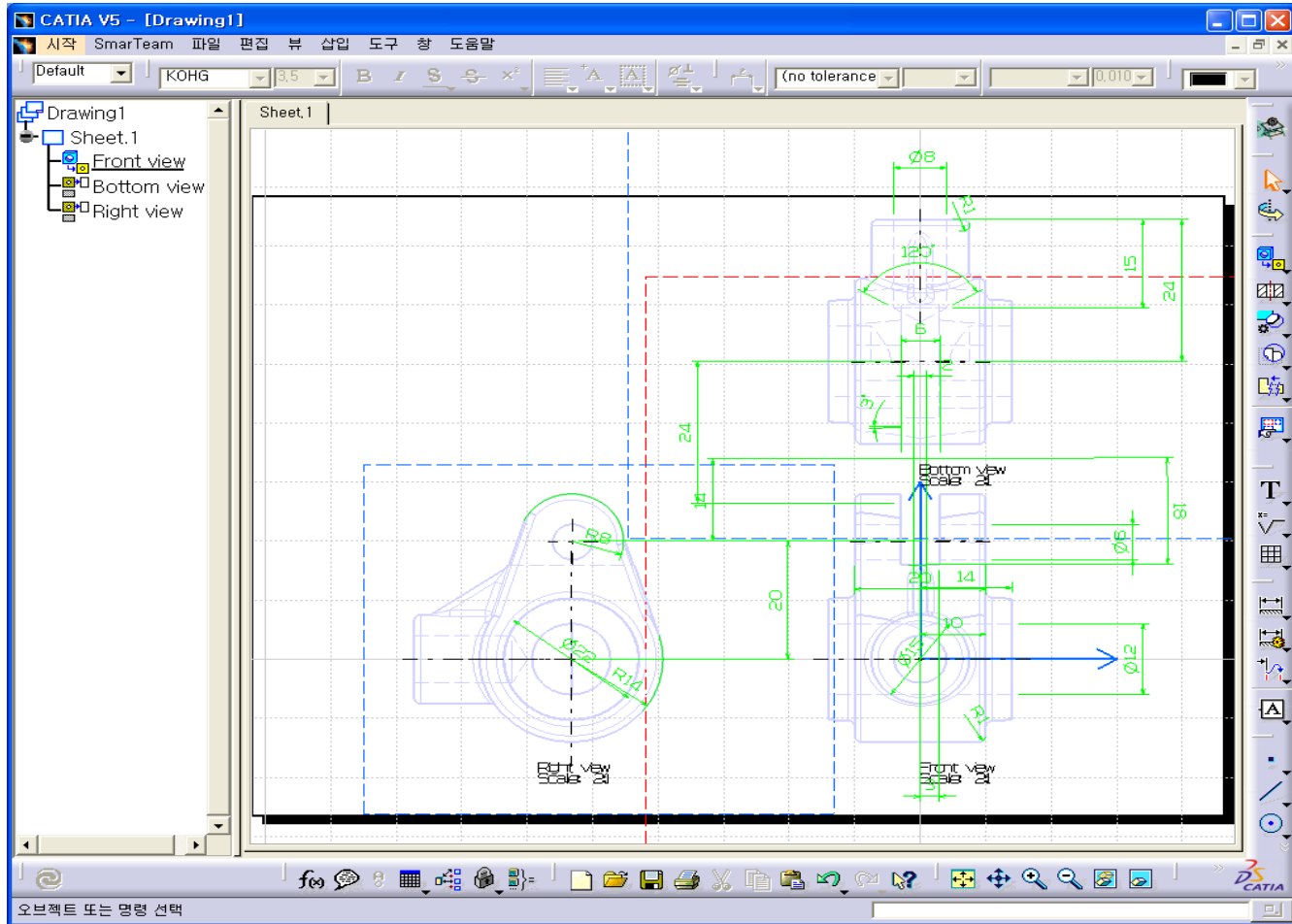
VII. DRAFTING

2. 단일 부품 드래프팅 (1)



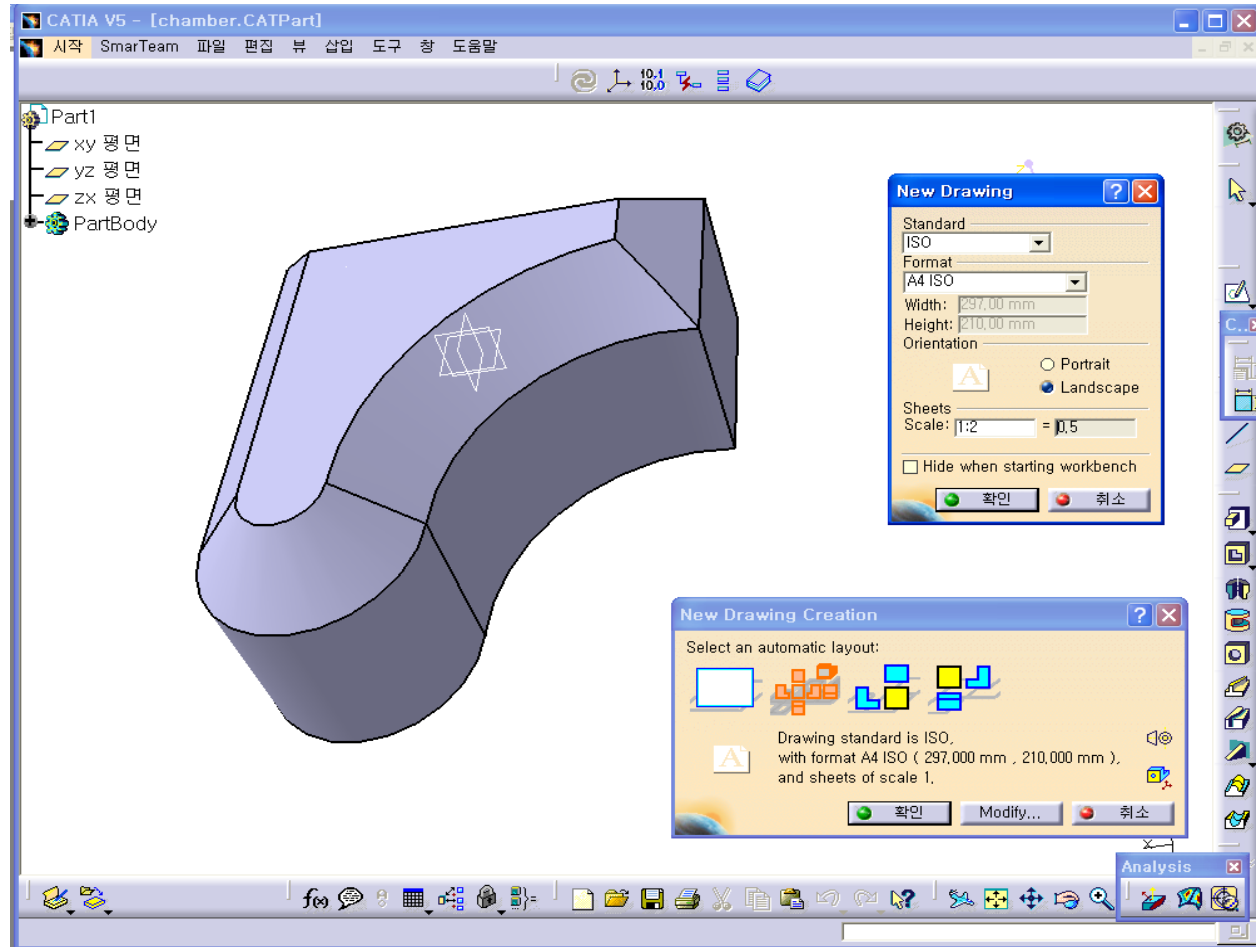
VII. DRAFTING

2. 단일 부품 드래프팅 (2)



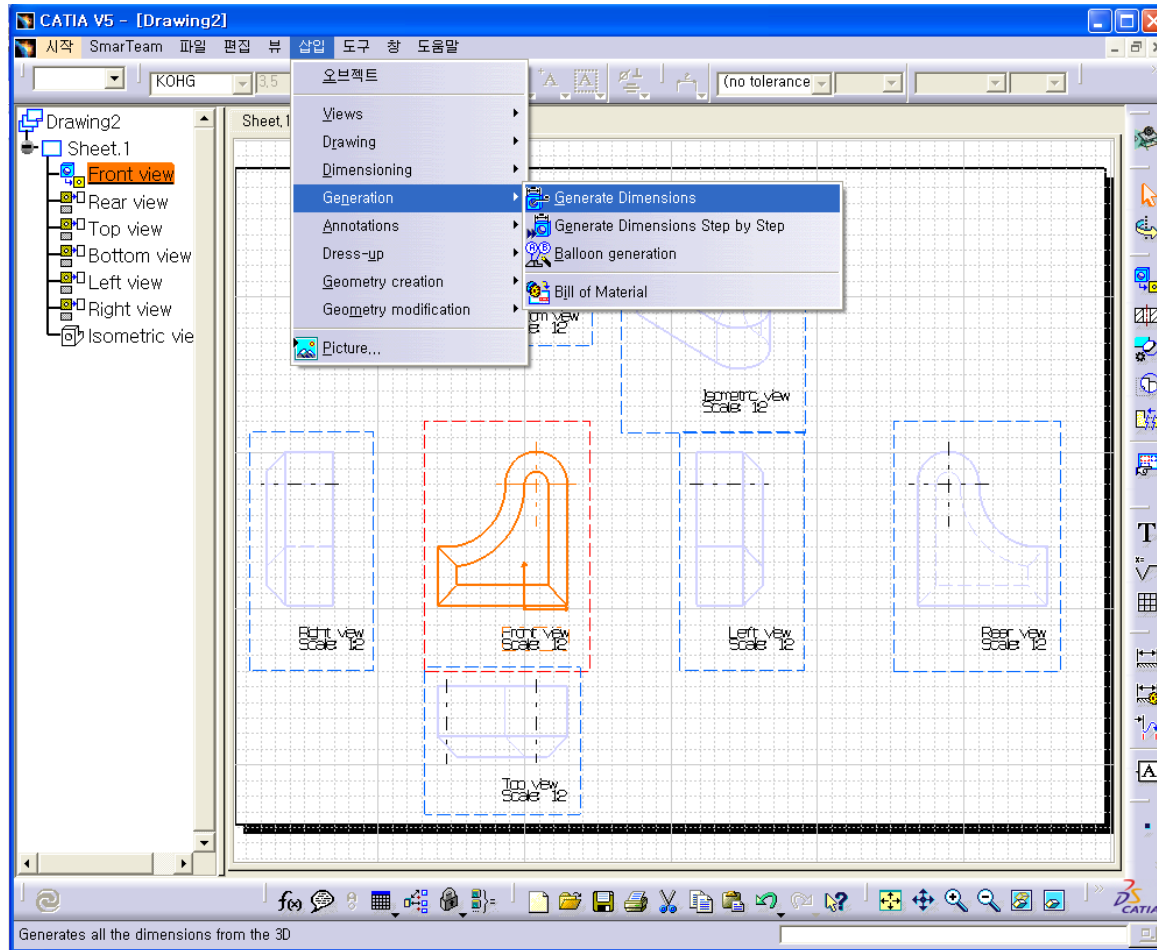
VII. DRAFTING

2. 단일 부품 드래프팅 (3)



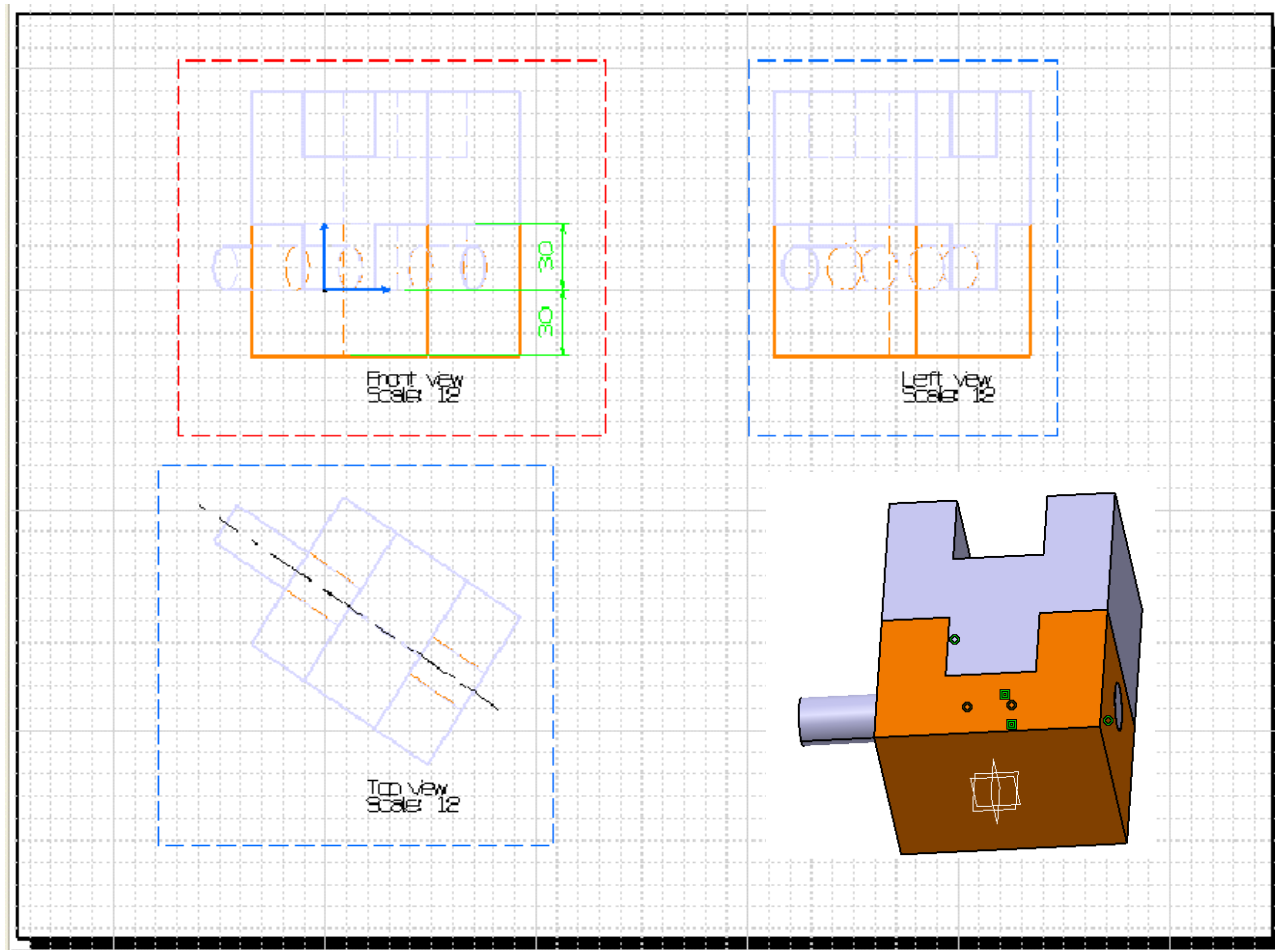
VII. DRAFTING

2. 단일 부품 드래프팅 (4)



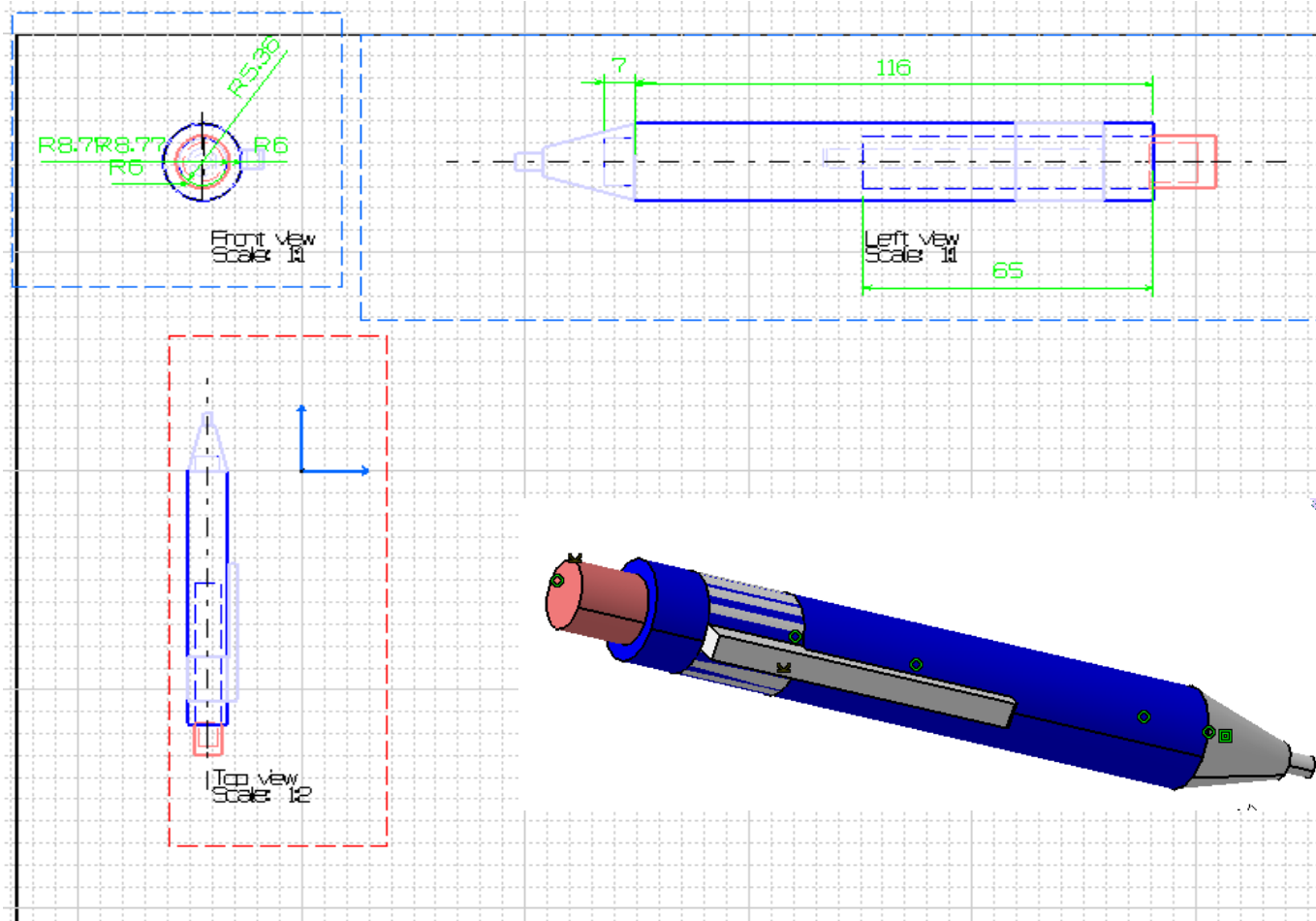
VII. DRAFTING

3. 조립품 드래프팅 (1)



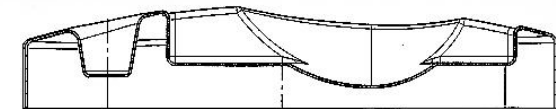
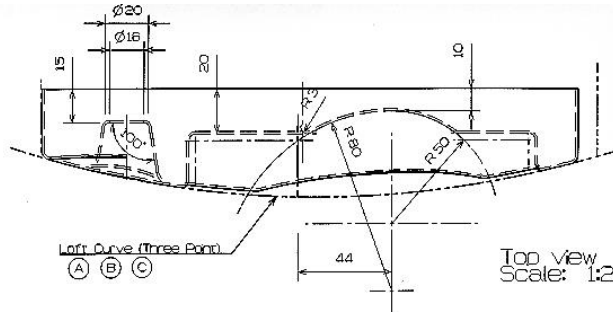
VII. DRAFTING

3. 조립품 드래프팅 (2)



VII. DRAFTING

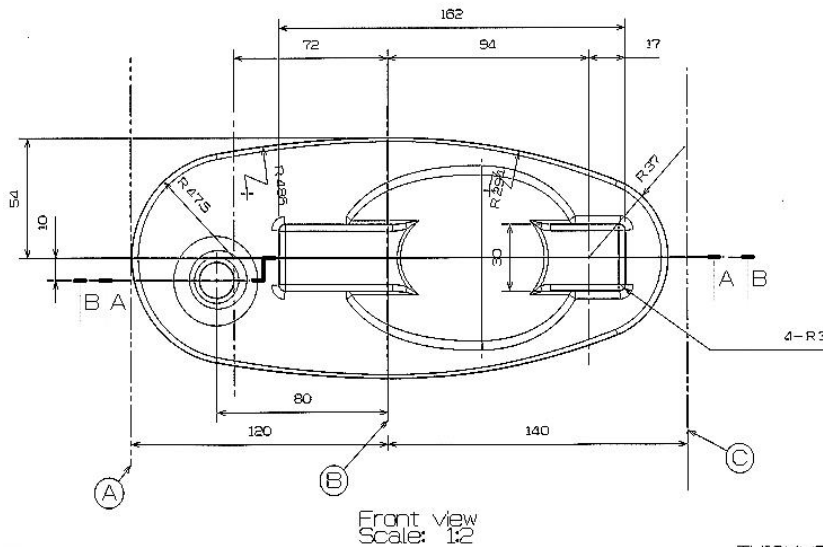
4. 드래프팅 예 제 (1)



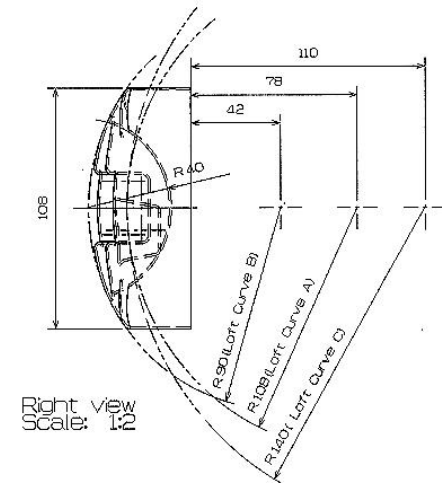
Section view A-A
Scale: 1:2



Section cut B-B
Scale: 1:2



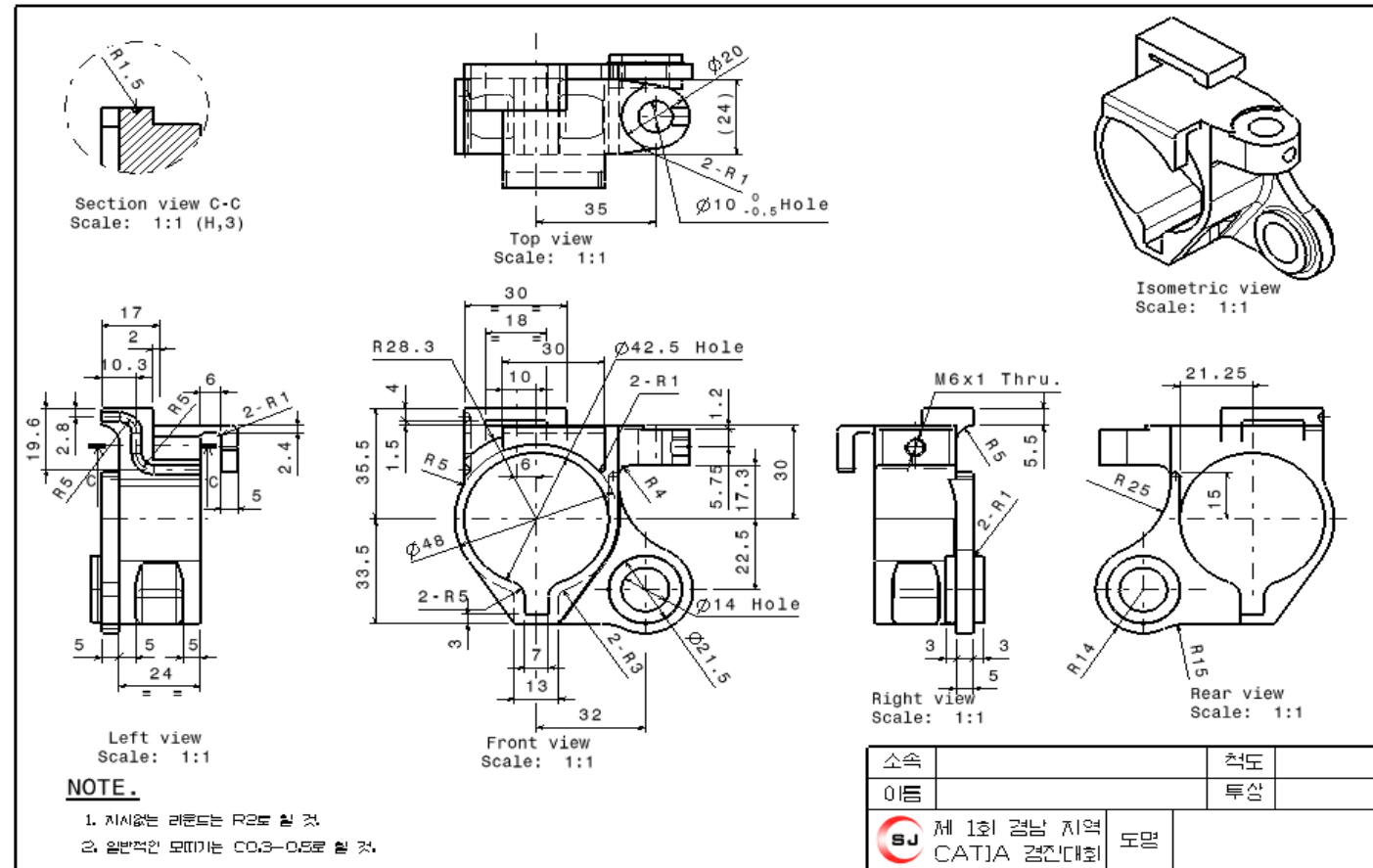
THICKNESS : 1mm



VII. DRAFTING

4. 드래프팅 예 제 (2)

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