

DRAFTING

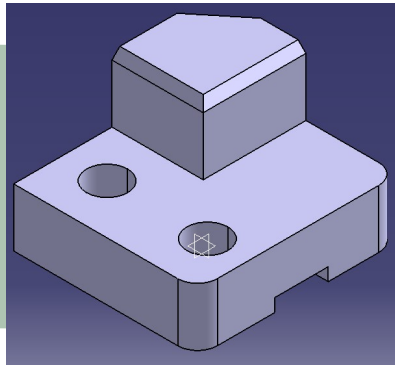
Computational Design Laboratory
Department of Automotive Engineering
Hanyang University, Seoul, Korea



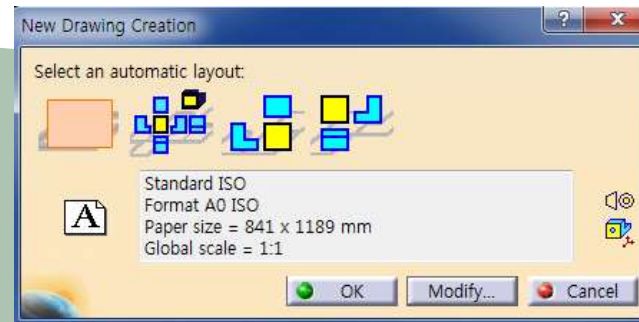
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- ✓ Dimensions
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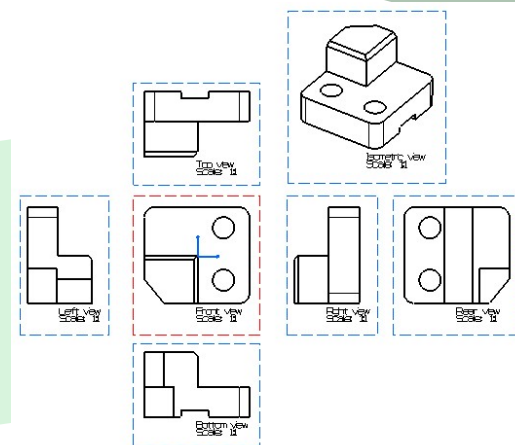
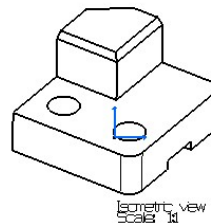
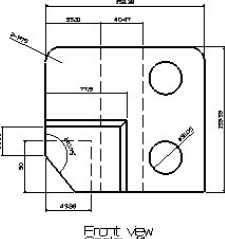
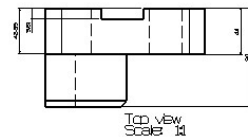
DRAFTING



모델 제작



표준 규격 및 용지크기 설정

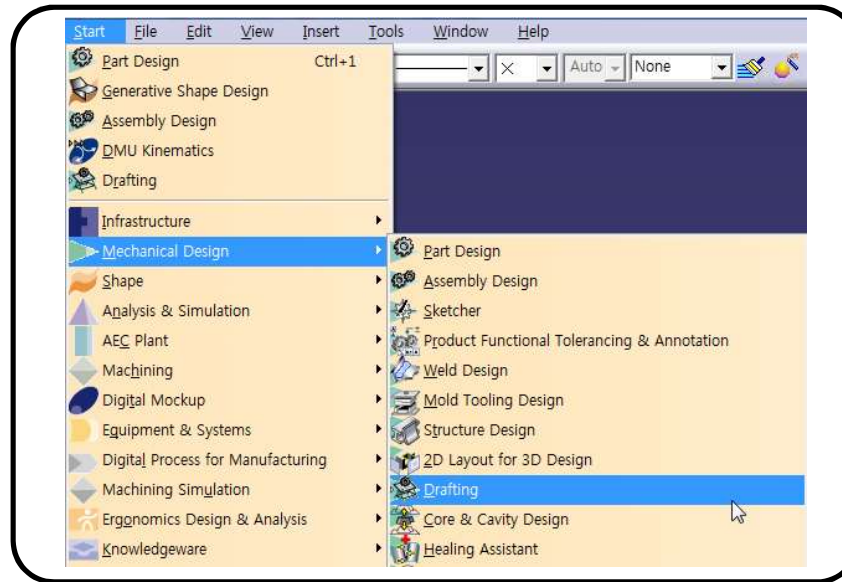


배치 확인

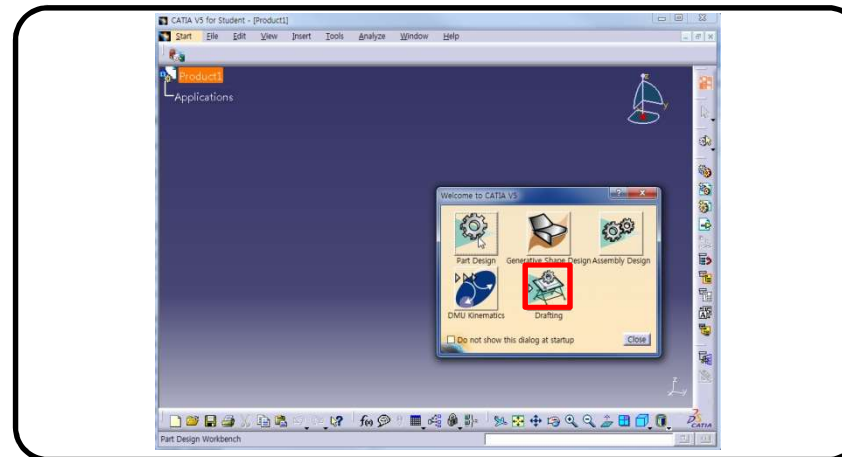
필요없는 투상도 제거 / 치수 입력

DRAFTING 시작하기 (1)

Start 메뉴에서 Mechanical Design → Drafting 선택

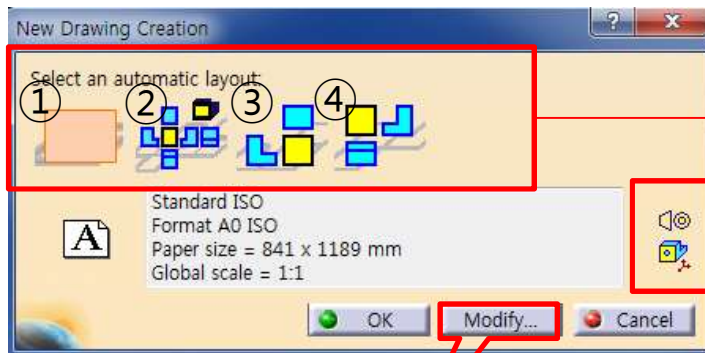


혹은, 카티아 실행화면에서 선택 (Start menu 설정시)



DRAFTING 시작하기 (2)

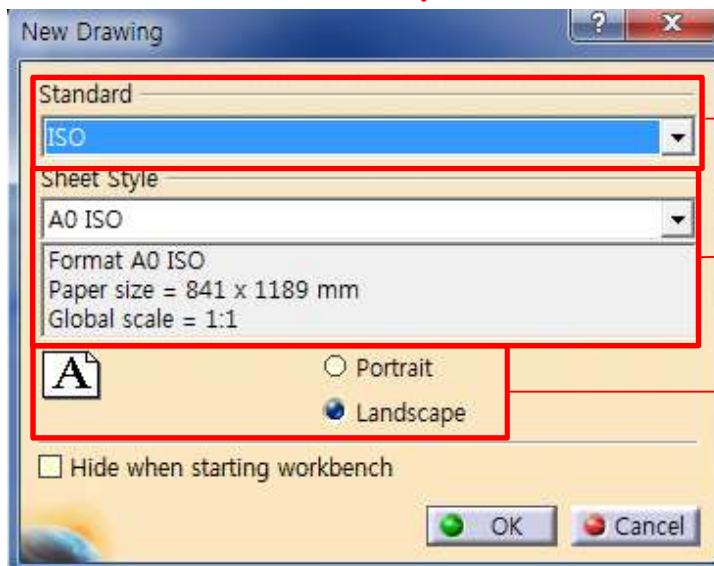
Drafting Workbench 실행 옵션 – ISO 규격/ Empty Sheet 선택



도면 배치 방법

- ① Empty Sheet
- ② All view
- ③ Front, Bottom and Right view
- ④ Front, Top and Left view

기본설정은 ISO - 1각법으로 되어있음

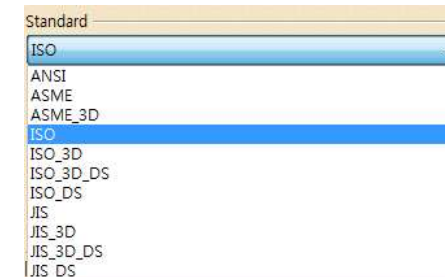


규격 종류

용지 크기

용지 방향

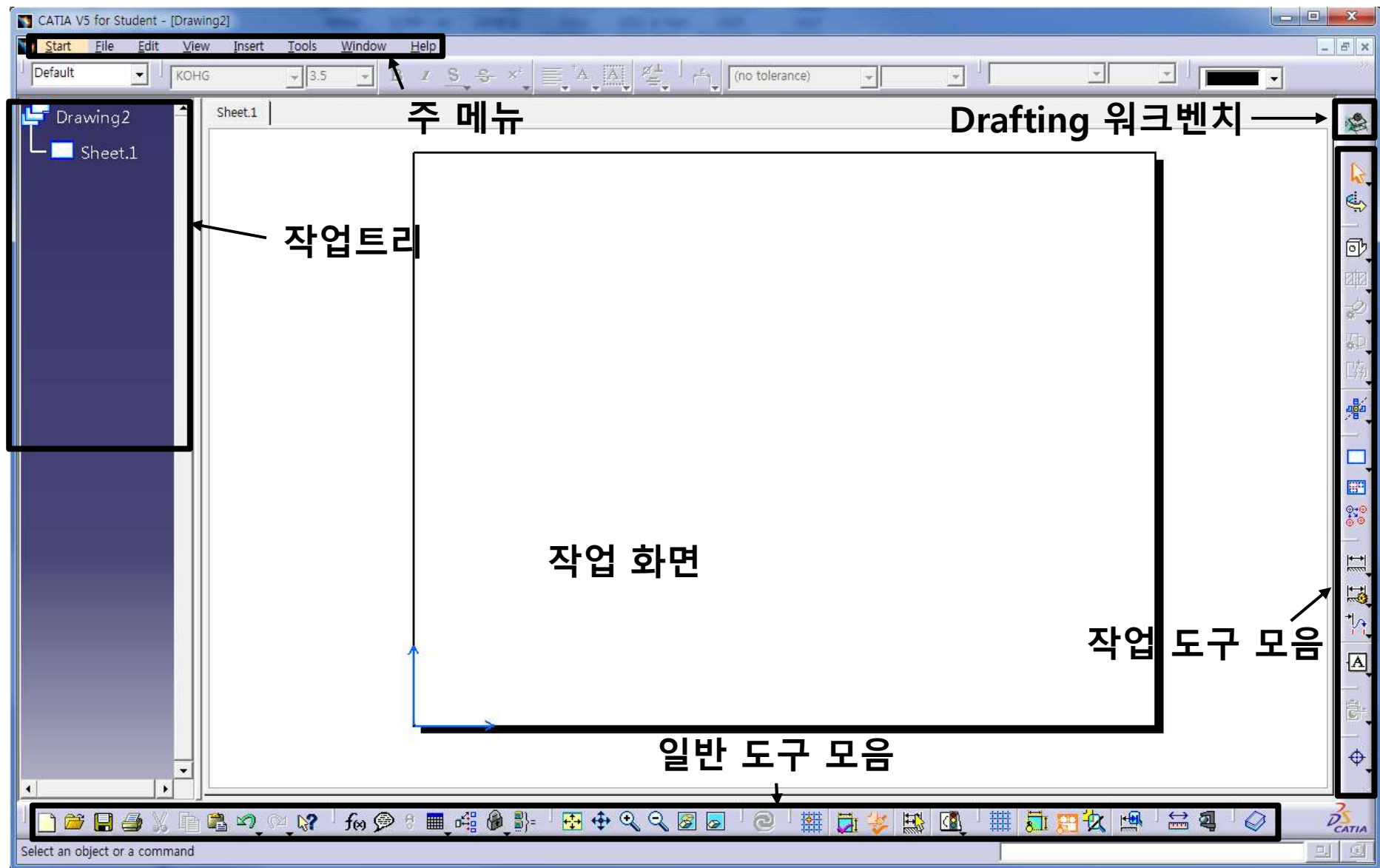
규격 종류



용지 크기



DRAFTING 작업화면 – EMPTY SHEET



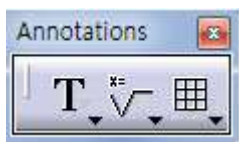
DRAFTING 툴바 종류



3차원 모델을 도면화 시키는 기능



도면에 치수를 입력



Text, Symbol, Table 등을 생성

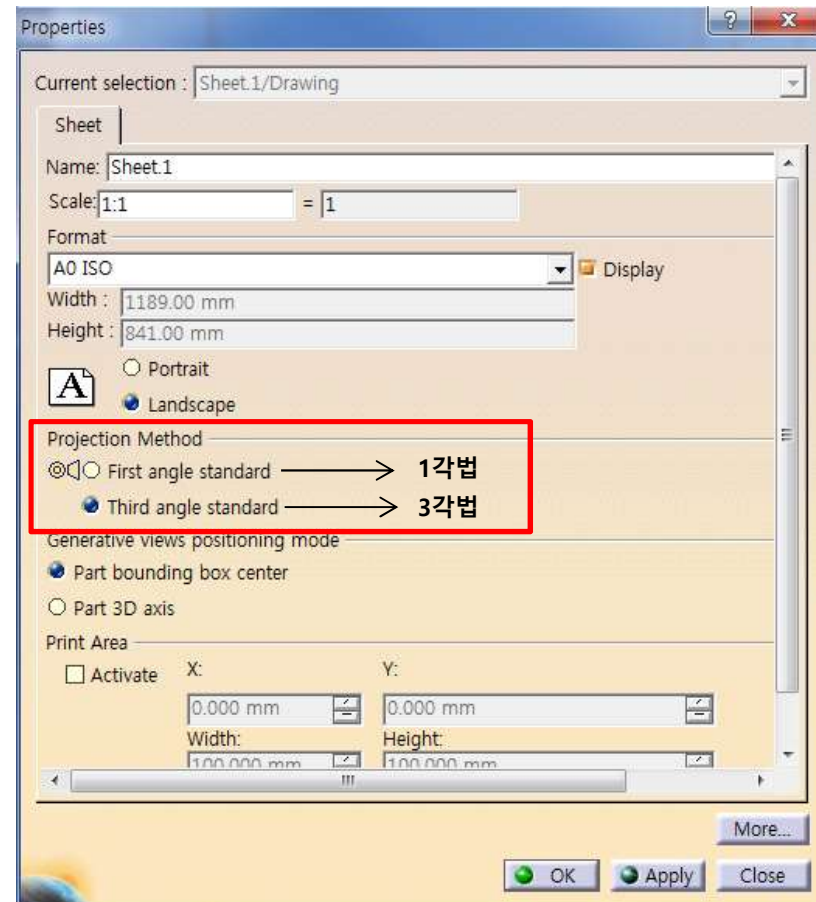


원 Profile 에 중심선 생성/ 해칭선 / 화살표 생성

3각법을 이용한 투상도 배치 (1)

ISO 규격 - Sheet Properties에서 1각법을 3각법으로 변경

- ISO 표준으로 실행하면 1각법으로 되어있으므로 3각법으로 변경하여 투상도 생성.



3각법을 이용한 투상도 배치 (2)

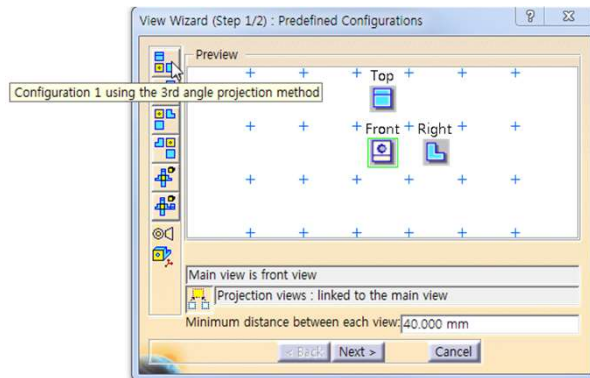
사용자가 원하는 투상도 생성



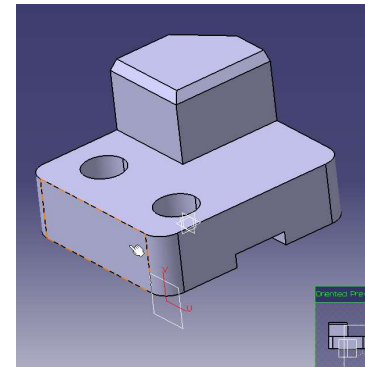
View Creation Wizard

- View의 생성 및 배치를 도와줌

① 기본 배치 설정

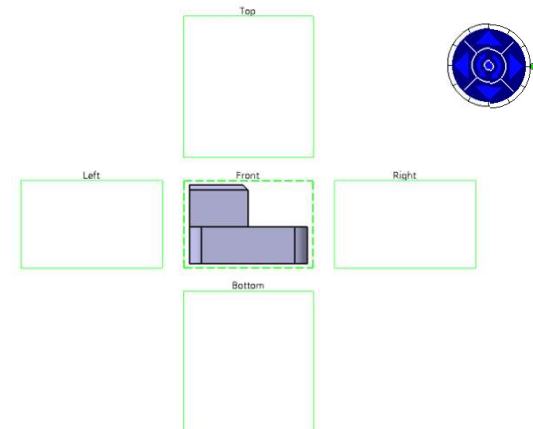
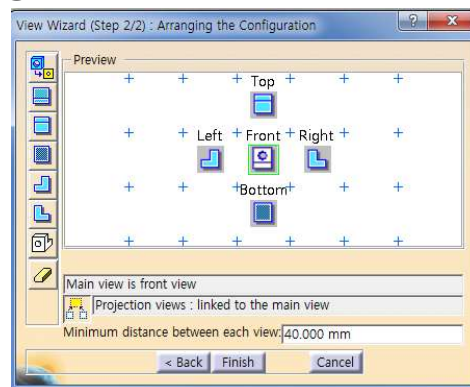


③ Front view로 설정할 면 선택



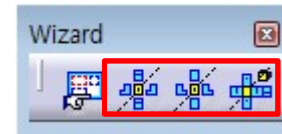
④ 우측 상단의 툴을 이용하여 Front view 최종 결정 후 배치

② 추가 배치 설정



3각법을 이용한 투상도 배치 (2)

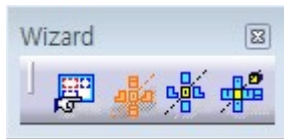
사용자가 원하는 투상도 생성



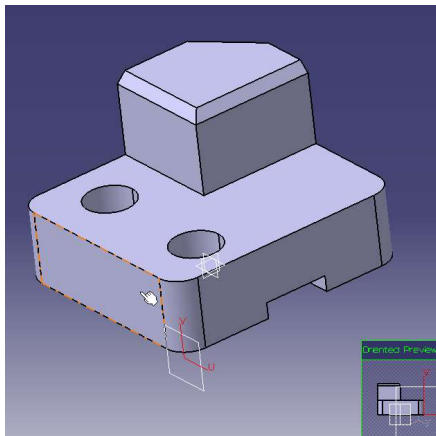
Front, Top and Left / Front, Bottom and Right / All Views
- 투상도의 배치를 좀 더 간편하게 할 수 있음



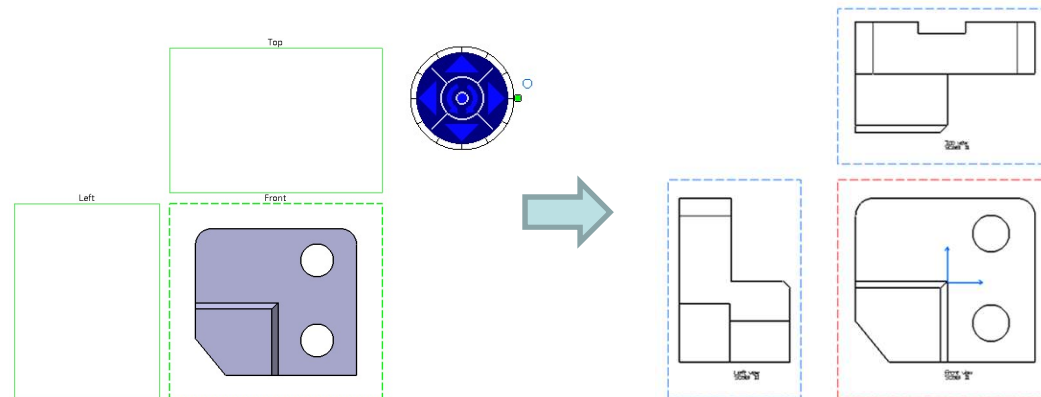
① 원하는 배치 선택



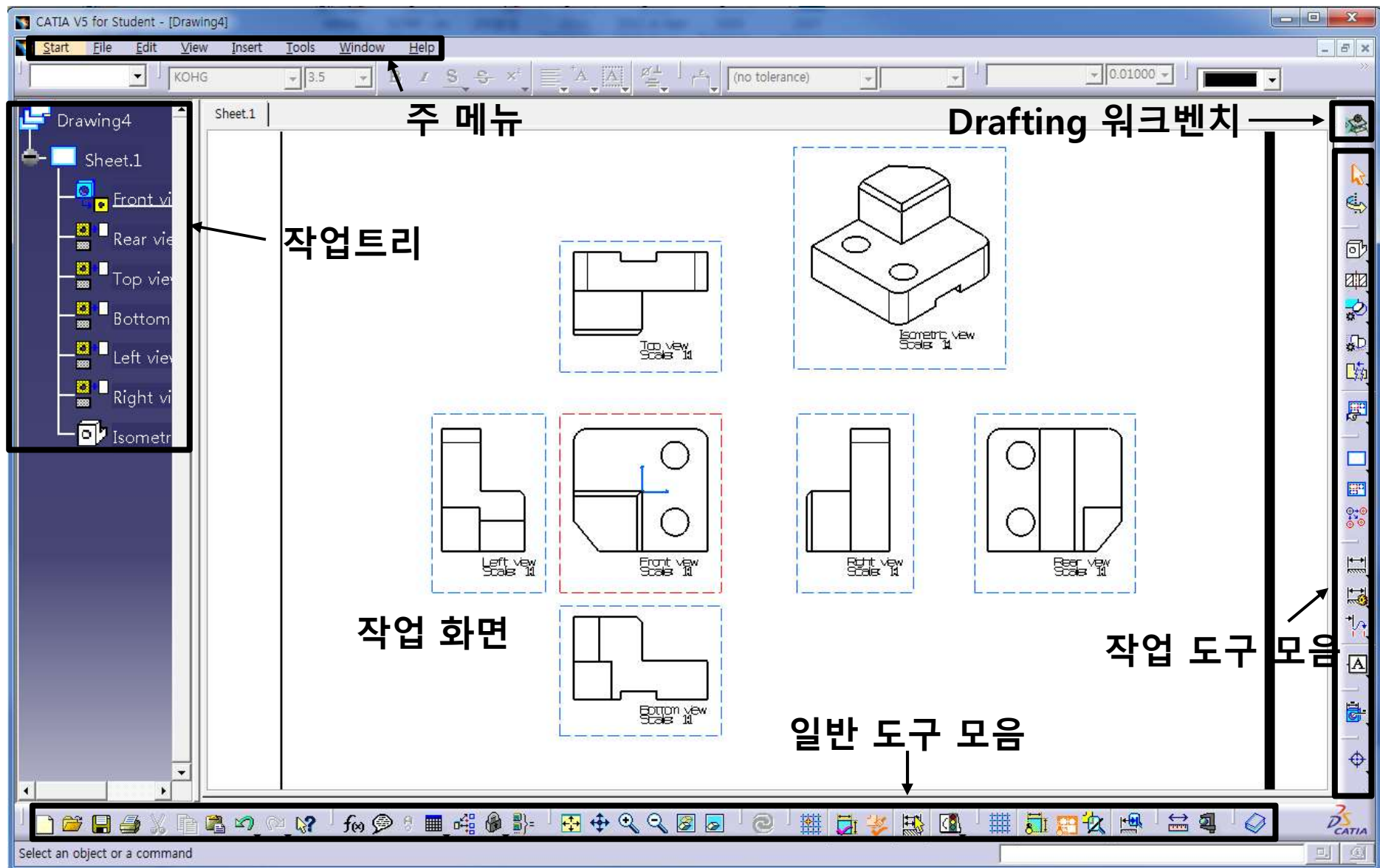
② Front view로 설정할 면 선택



③ 우측 상단의 툴을 이용하여 투상면의 위치 최종 결정 후 배치

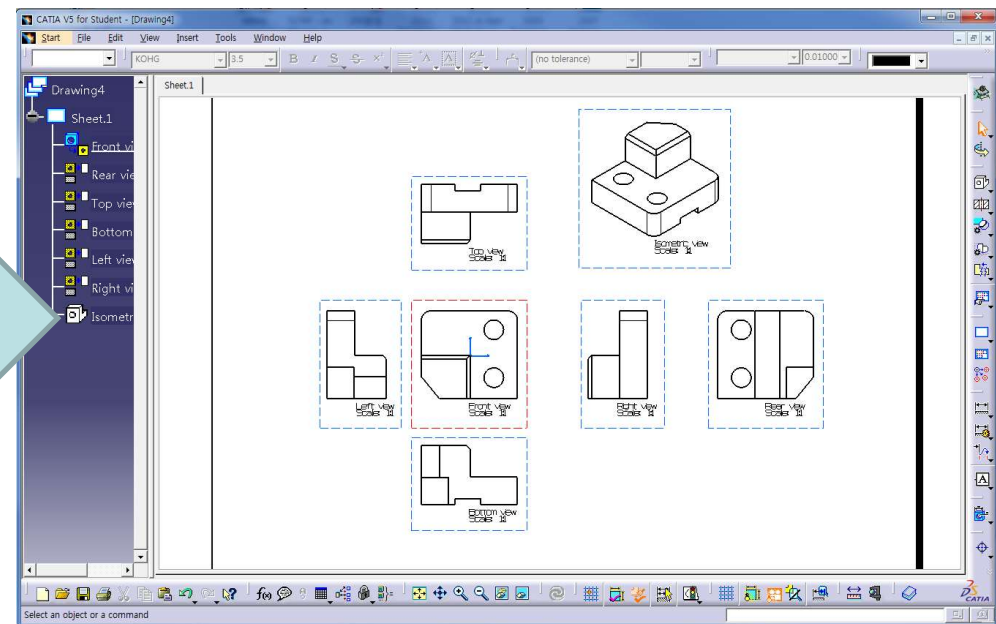
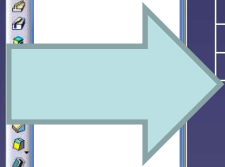
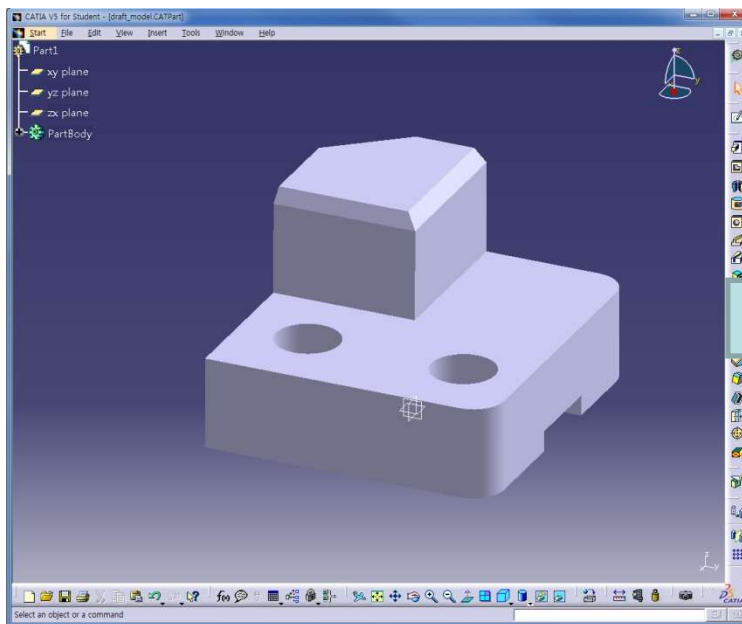


DRAFTING 작업화면



모델과 DRAFTING의 관계

모델이 변경 될 경우 화면 하단의 Update 버튼을 클릭
하면 변경된 모델의 정보가 도면에 적용된다.

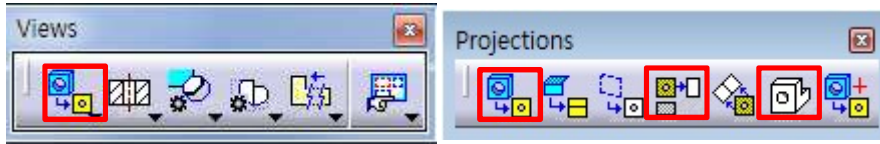


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DRAFTING TOOLS

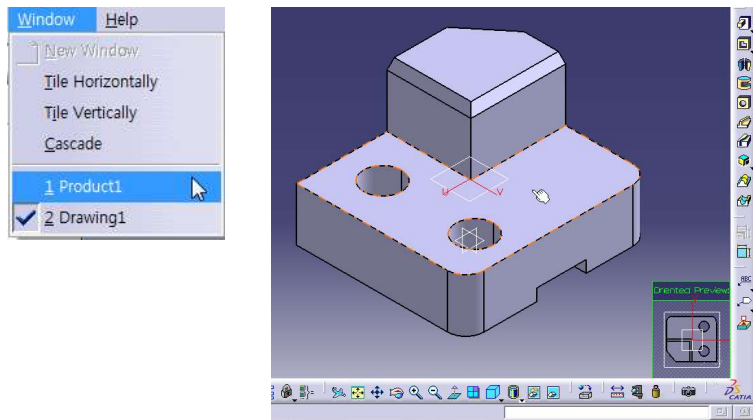
Views



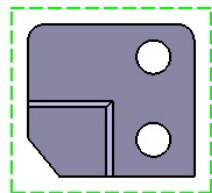
Front View

- 원하는 위치를 front view로 생성

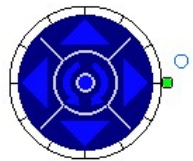
- 아이콘 클릭 후 모델화면으로 전환하여 원하는 면 클릭



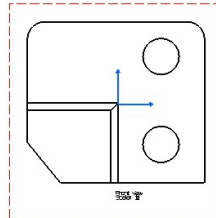
도면에 1차 배치



회전

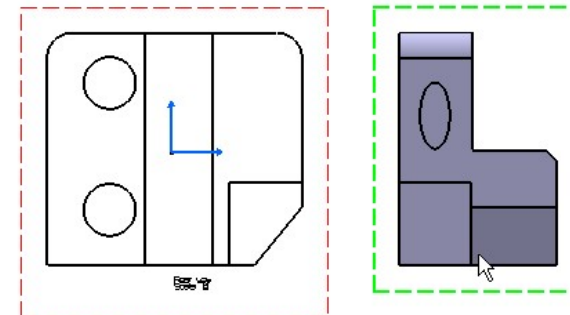


최종 배치



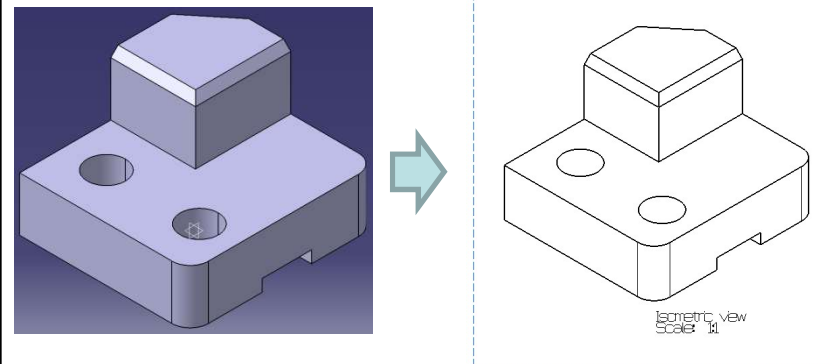
Projection View

- 활성화된 뷰에 대한 projection view 생성



Isometric View

- 모델의 isometric view 생성



DRAFTING TOOLS

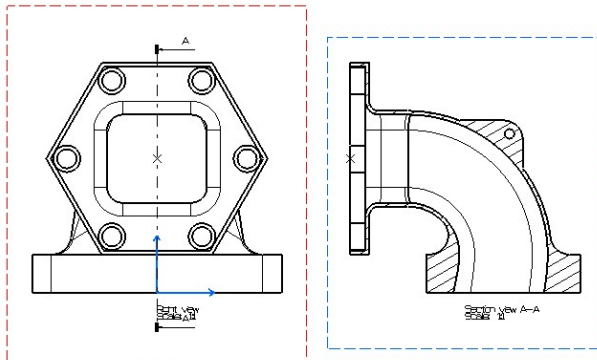
Views



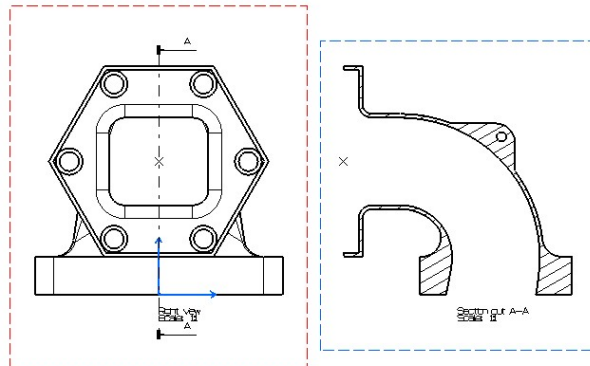
Offset Section View /Cut

- View : Line을 기준으로 잘려진 면에서 바라본 Solid 모델의 단면도를 보여줌
- Cut : 내부 단면도의 잘려진 부분만 보여줌

Offset Section View



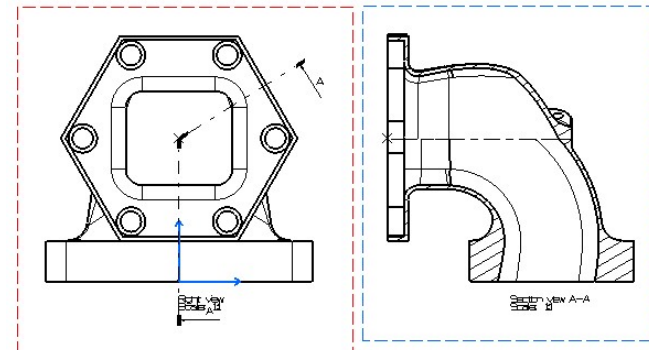
Offset Section Cut



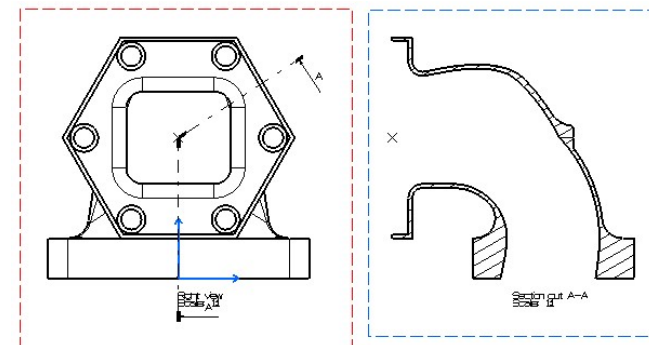
Aligned Section View /Cut

- View : 연속된 Line을 기준으로 잘려진 면에서 바라본 Solid 모델의 단면도를 보여줌
- Cut : 내부 단면도의 잘려진 부분만 보여줌

Aligned Section View

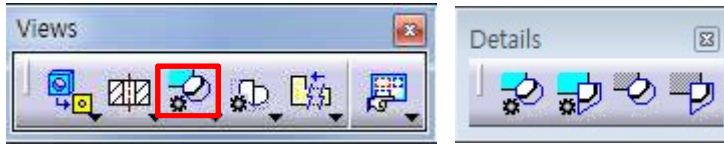


Aligned Section Cut



DRAFTING TOOLS

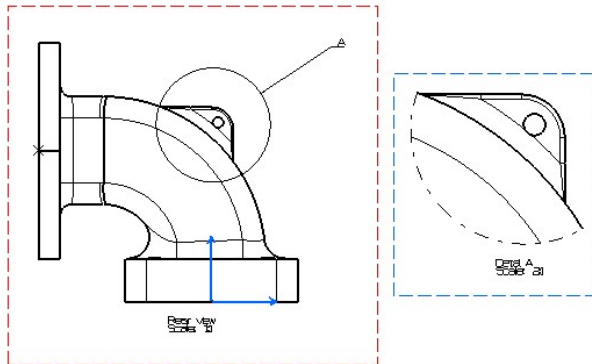
Views



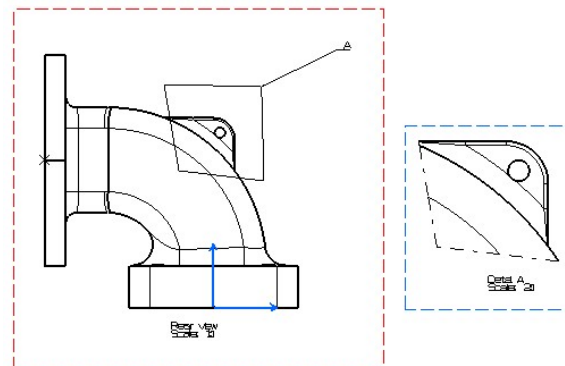
Detail View  / Detail View Profile 

- Detail View : 원형 profile의 부분을 확대해서 보여줌
- Detail View Profile : 사용자 정의의 profile의 부분을 확대해서 보여줌

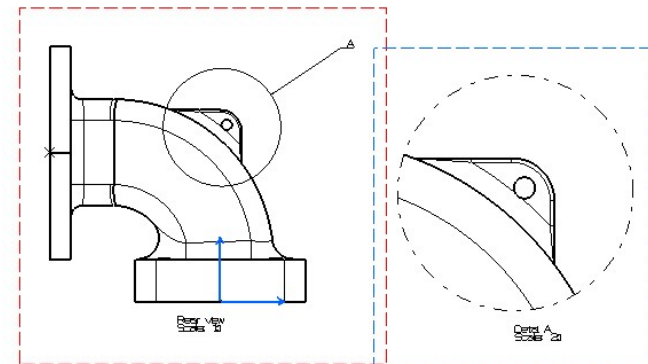
Detail View



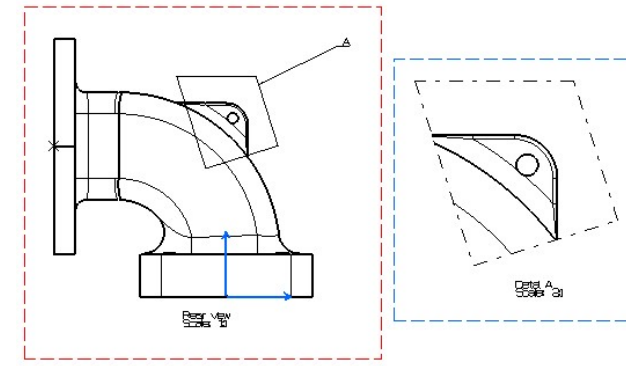
Detail View Profile



Quick Detail View



Quick Detail View Profile



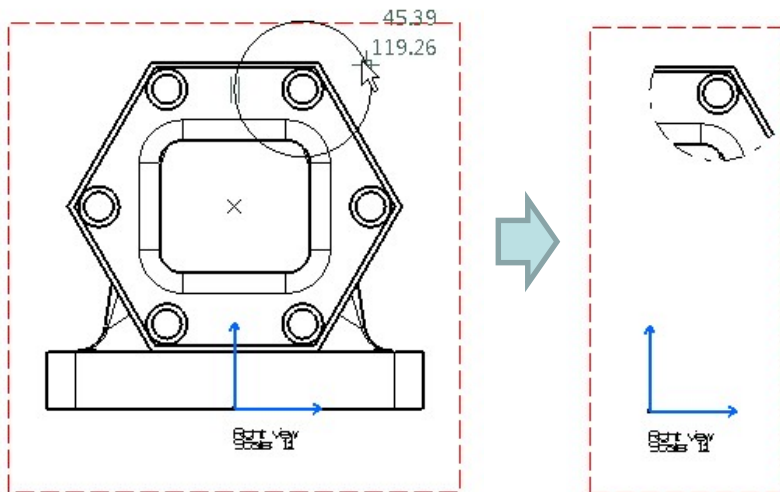
DRAFTING TOOLS

Views



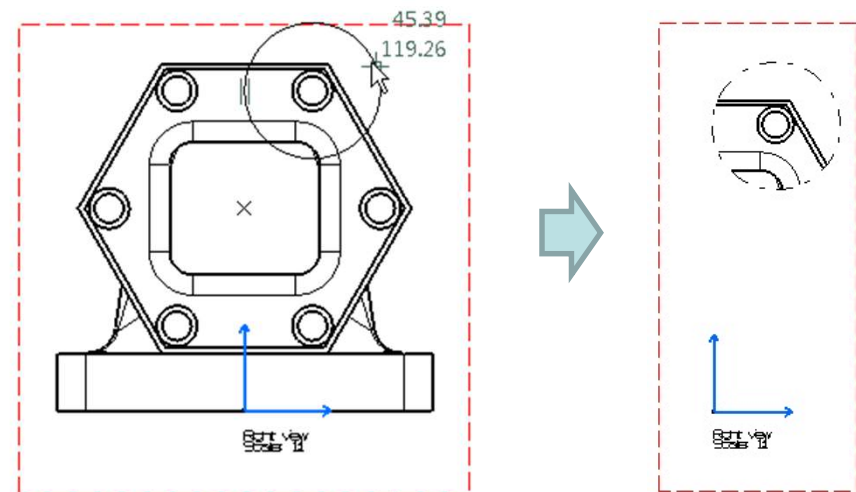
Clipping View  / Clipping View Profile 

- Clipping: 선택한 부분만 남기고 제거



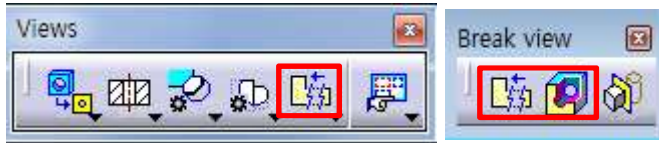
Quick Clipping View  / Quick Clipping View Profile 

- Clipping view와는 다르게 2D 도면 정보를 기준으로 진행



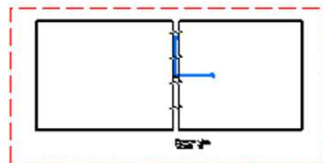
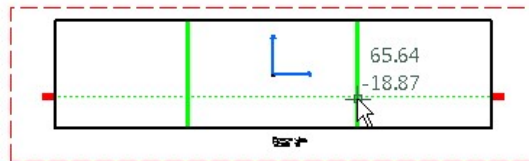
DRAFTING TOOLS

Views



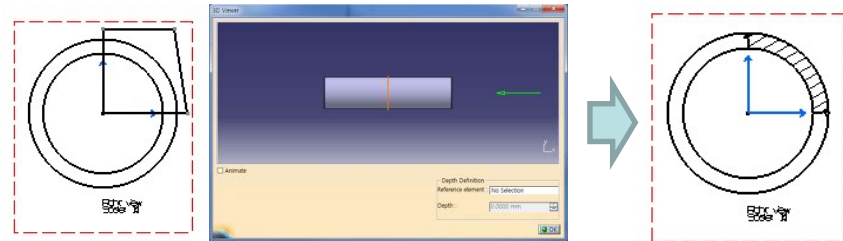
Broken View

- 반복되는 형상이나 너무 긴 도면의 일부분을 생략하여 표현해주는 기능



Break View

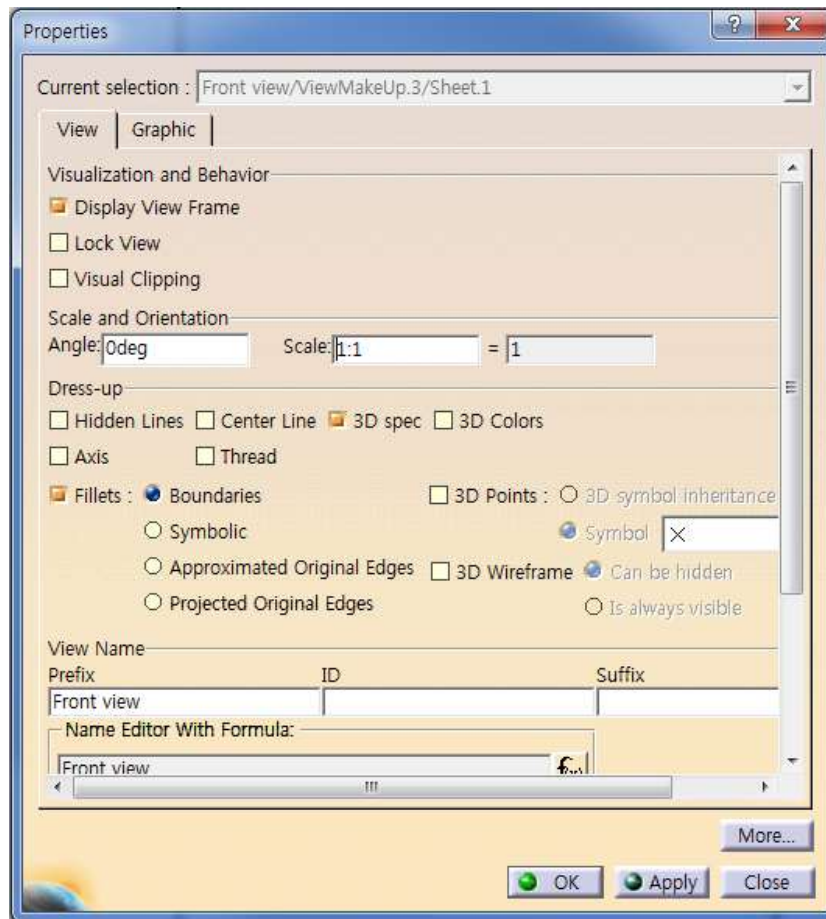
- 단면의 선택부분을 잘라서 내부 단면을 표시



DRAFTING TOOLS

View Properties

원하는 투상도에 마우스 오른쪽버튼을 클릭하여 properties 선택



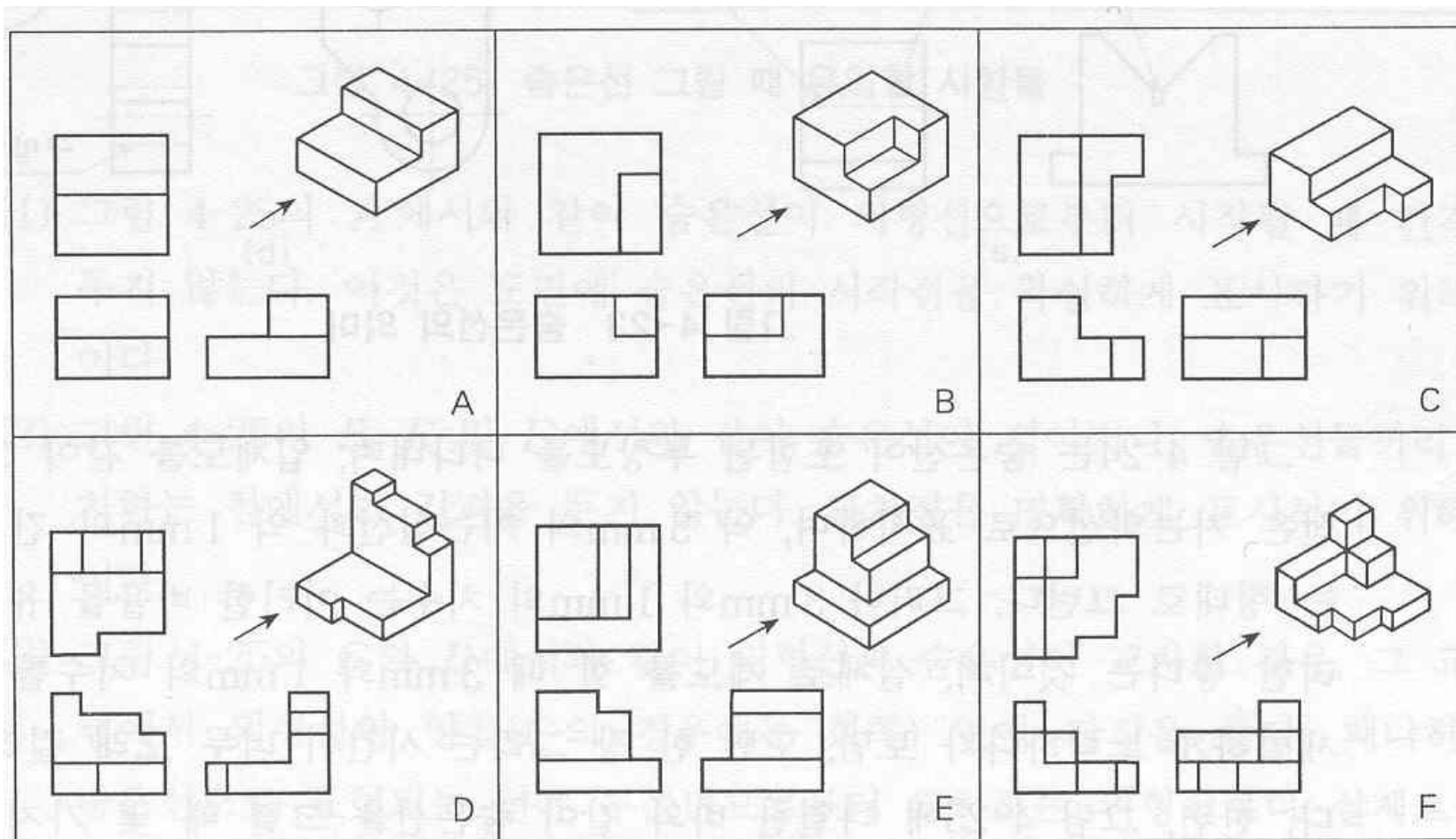
Hidden Lines : 숨은선 표시

Center Line : 원형 도형에 중심선 표시

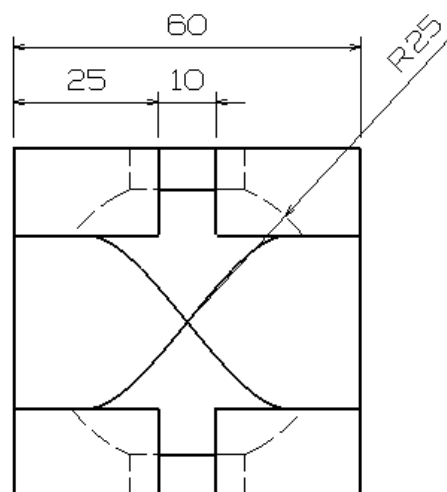
Axis : 회전체의 축 표시

Fillets : 필렛으로 인한 보조선 표시

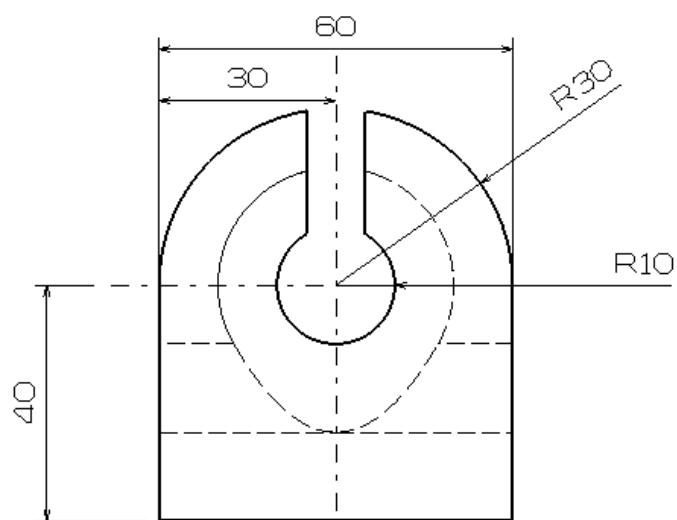
투상도를 적용한 예



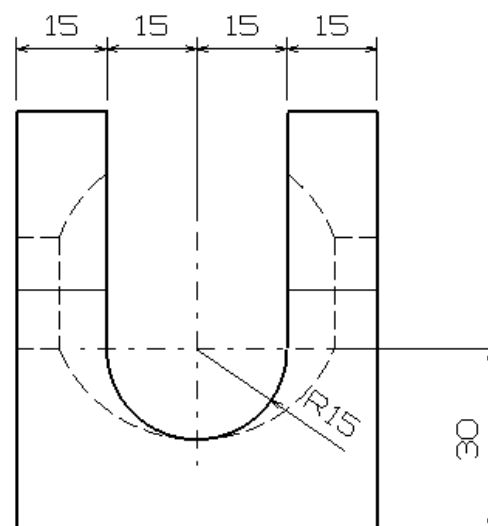
실습 예제



Top view
Scale: 1:1



Front view
Scale: 1:1



Right view
Scale: 1:1

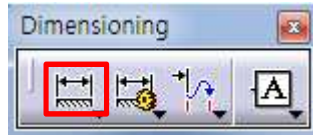
다음의 도면을 참고하여 모델링 한 후
도면과 같이 배치하시오. (숨은선 포함)

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Dimensioning



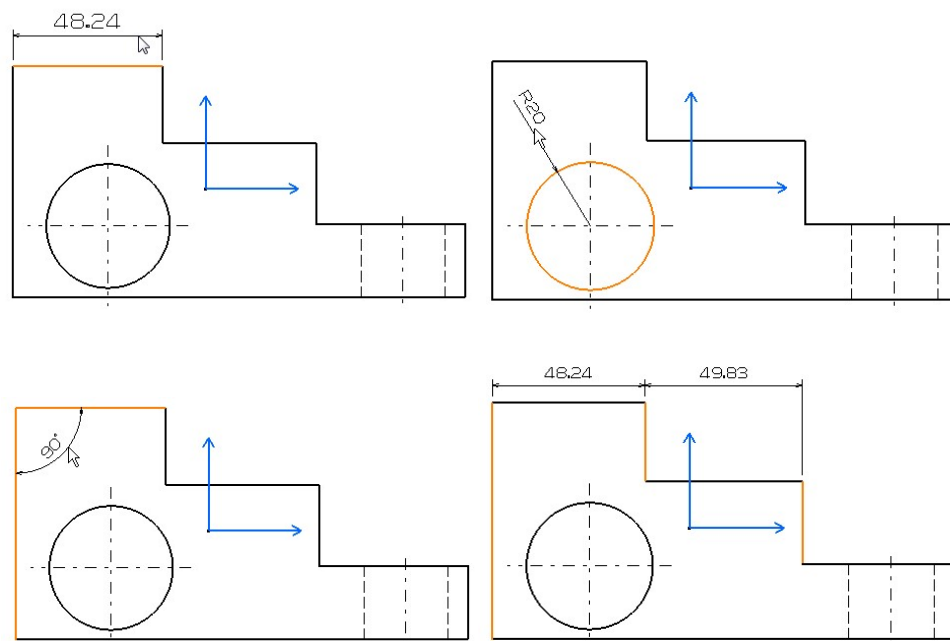
-  Dimensions : 도면에 치수를 생성
-  Chained Dimensions : 기준 Line에서 연속적으로 치수를 생성
-  Cumulated Dimensions : 기준 Line에서 누적된 치수를 생성
-  Stacked Dimensions : 계단식 치수를 생성
-  Length/Distance Dimensions : 길이 값과 거리 값을 나타냄
-  Angle Dimensions : 선택한 두 개의 Element의 각도를 나타냄
-  Radius Dimensions : 선택한 circle 또는 arc의 반지름 값을 나타냄
-  Diameter Dimensions : 선택한 circle 또는 arc의 직경 값을 나타냄
-  Chamfer Dimensions : 도면 중 chamfer 된 부분에 치수를 생성
-  Thread Dimensions : Thread가 적용된 홀이나 pocket에 thread 치수를 생성
-  Coordinate Dimensions : 2D 도면상의 좌표 치수를 생성
-  Coordinate Dimension Table : 도면의 좌표 값을 표시하는 치수를 table로 생성
-  Hole Dimension Table : Hole의 Center Point 좌표 값과 Diameter 값을 table로 생성

DRAFTING TOOLS

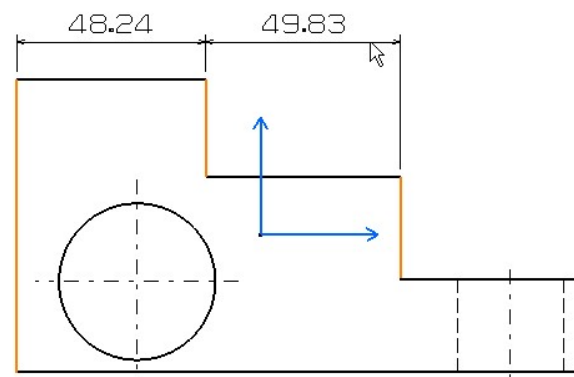
Dimensioning



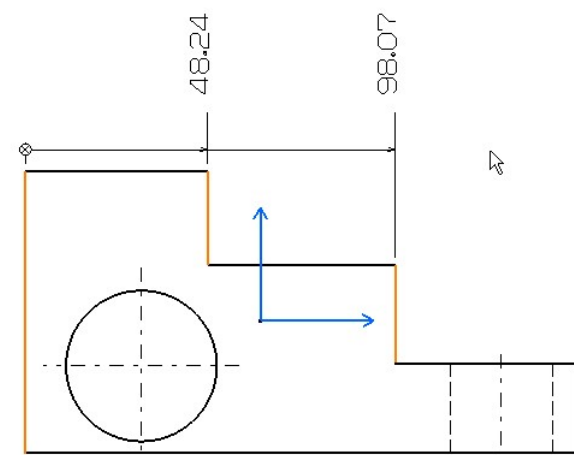
Dimensions



Chained Dimensions



Cumulated Dimensions

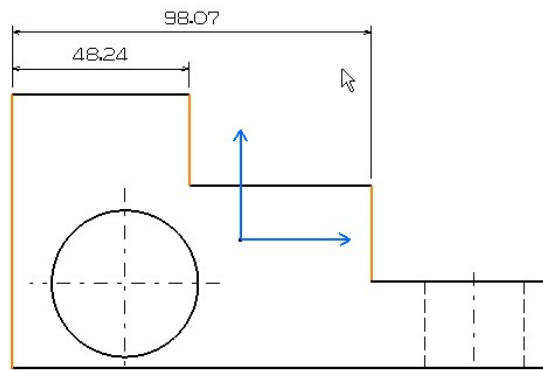


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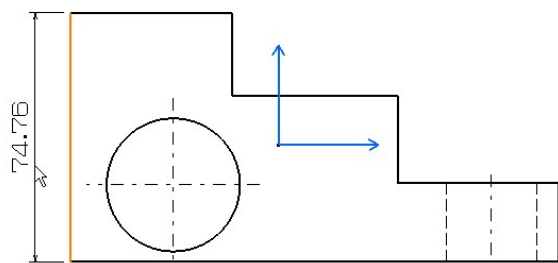
Dimensioning



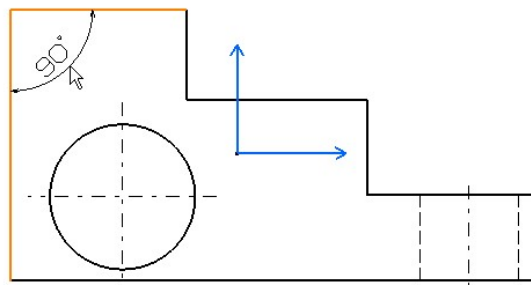
Stacked Dimensions



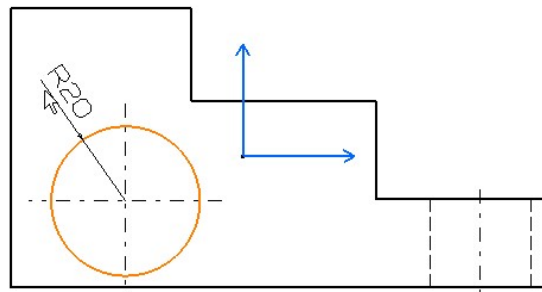
Length/Distance Dimensions



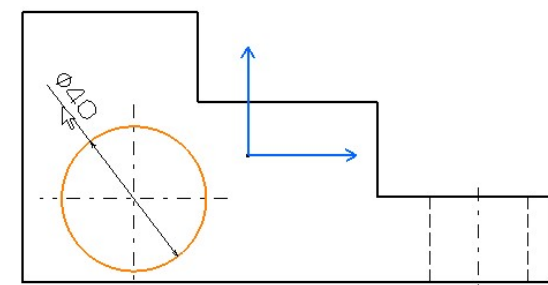
Angle Dimensions



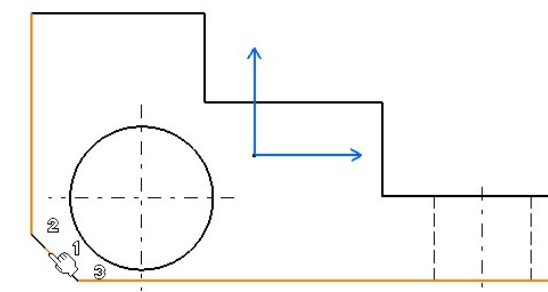
Radius Dimensions



Diameter Dimensions



Chamfer Dimensions



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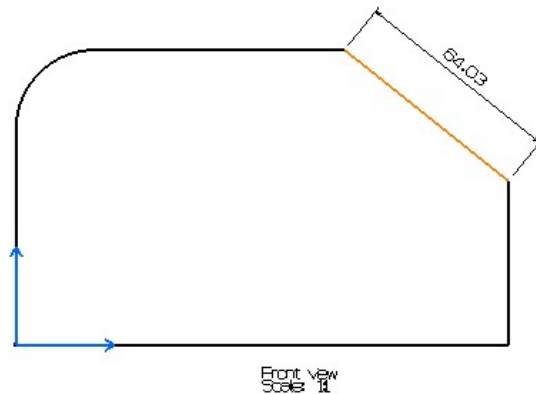
Tool palette



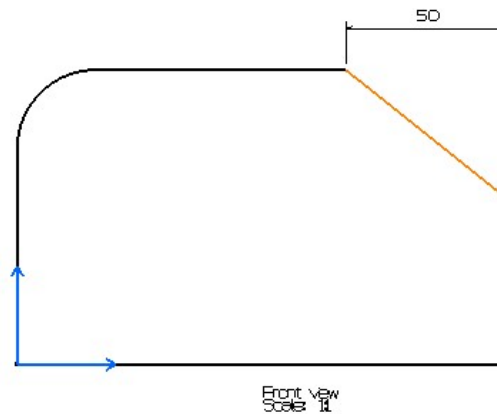
- 치수를 생성할 때 방향, 위치 등을 설정

-  Projected dimension : 마우스 위치에 따라 요소 치수, 수평 치수, 수직 치수로 설정
-  Dimension on element : 선택한 요소의 값을 측정
-  Horizontal dimension : 선택한 요소의 수평 방향의 값을 측정
-  Vertical dimension : 선택한 요소의 수직 방향의 값을 측정
-  Intersection point detection : 교차, 연장되는 위치에 가상의 점을 생성

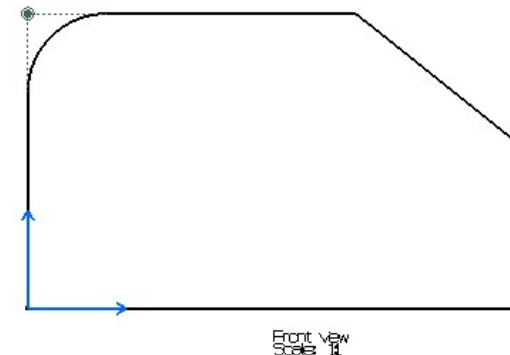
<Dimension on element>



<Horizontal dimension>



<Intersection point detection>



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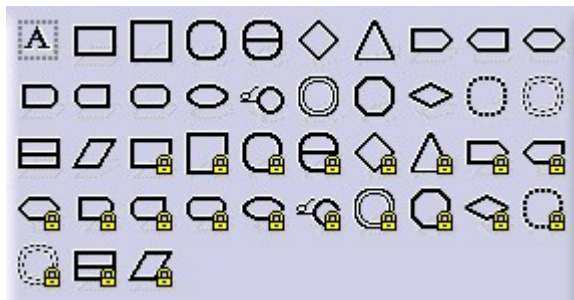
Text / Dimension Properties



도면에 입력되는 글씨의 크기, 종류 등을 조절

Frame

- 치수를 둘러싼 프레임 형태 조절



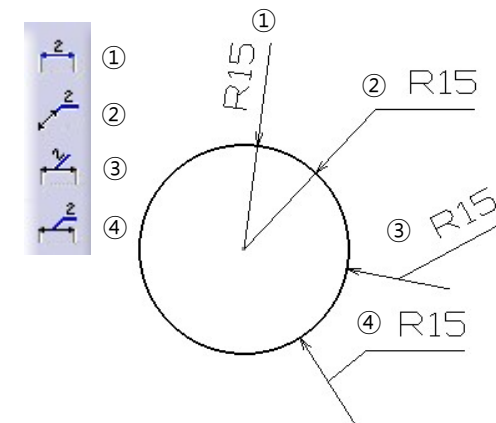
Insert Symbol

- 치수 외에 추가적인 심볼 생성



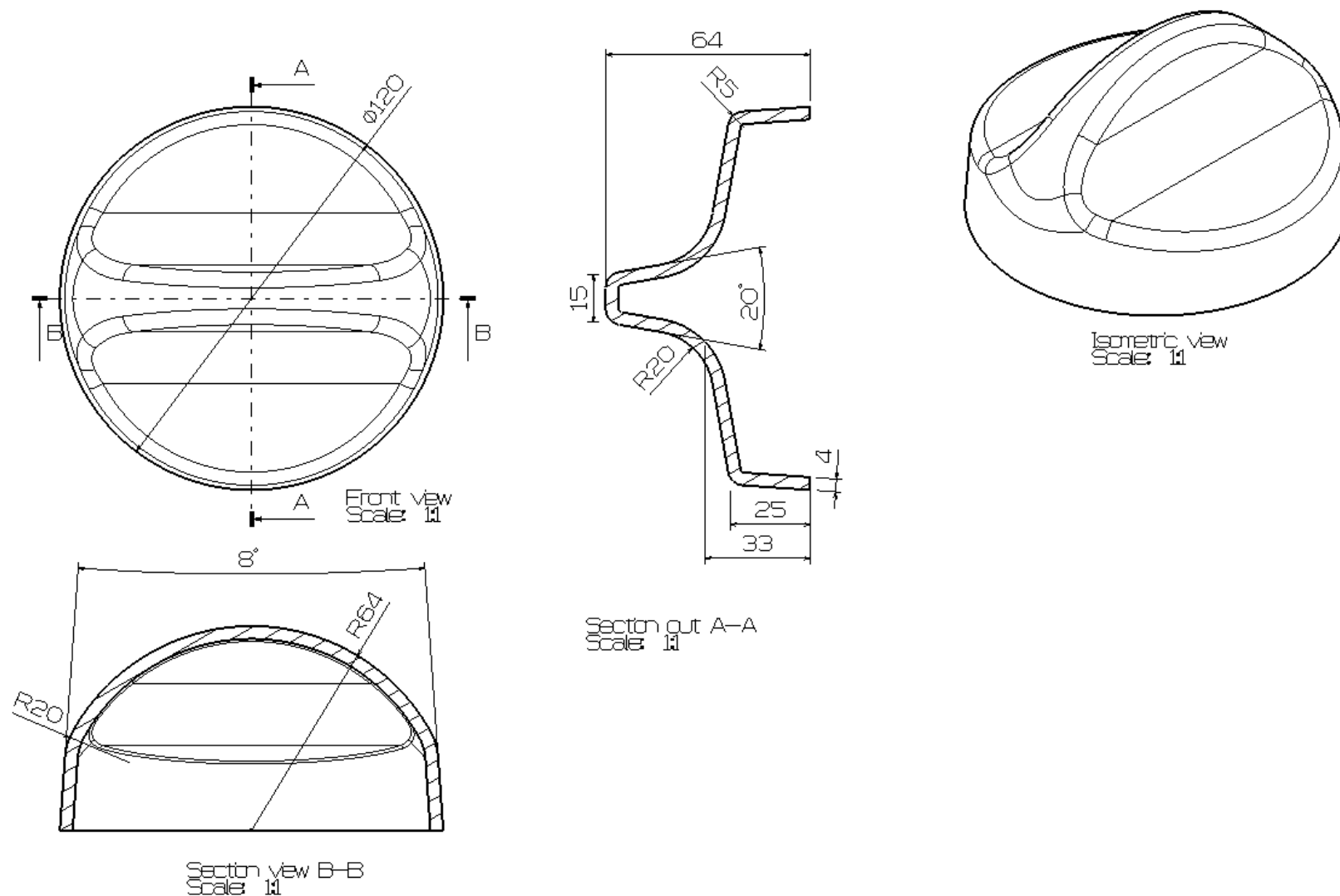
Dimension Line

- 치수를 입력하는 방식 선택



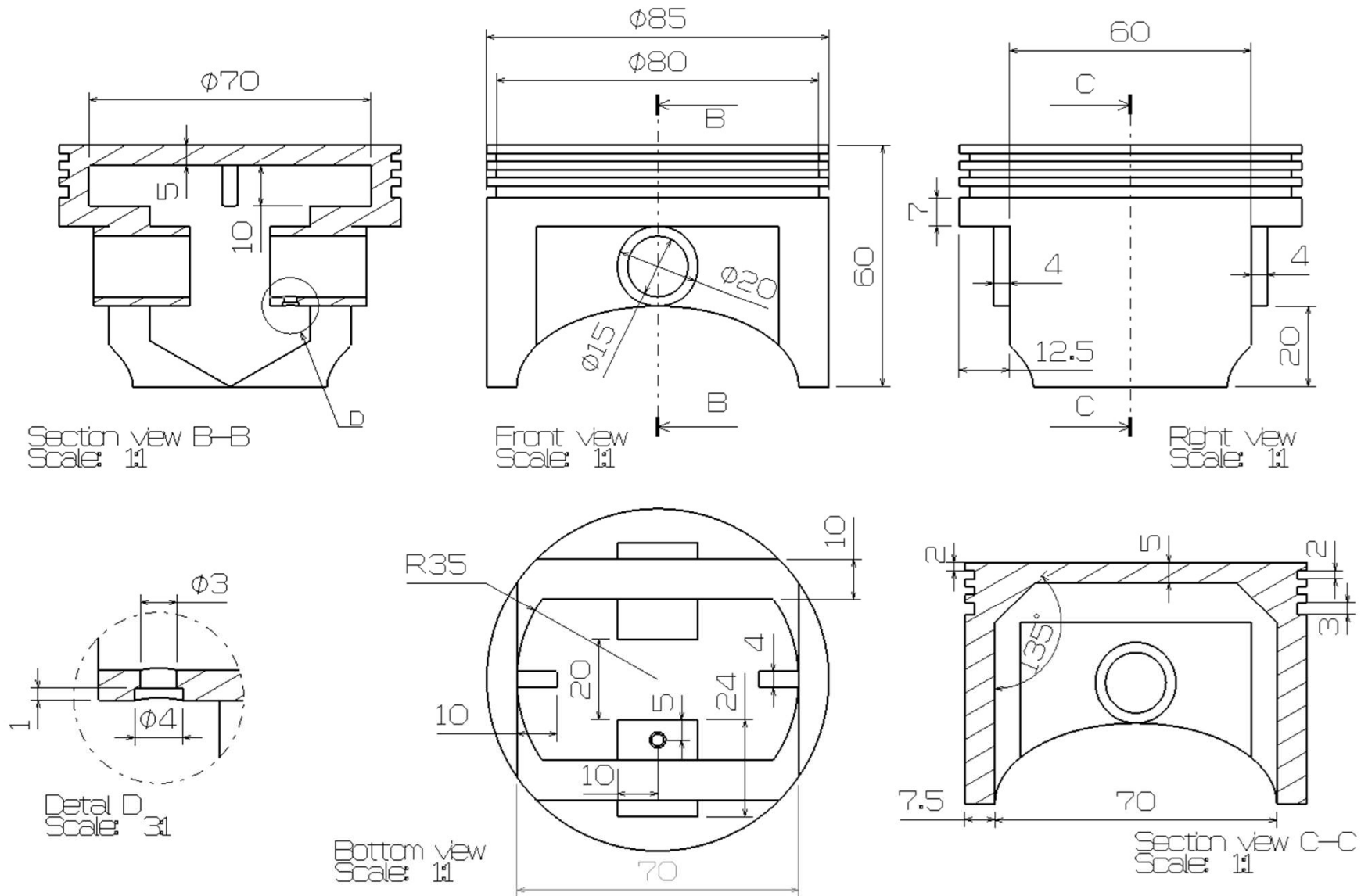
실습 예제

다음의 도면과 동일하게 투상도 생성 및 치수 입력 하기(Controller 모델 이용)



실습 예제

다음 도면을 보고 투상도 배치 및 치수 입력 해보기

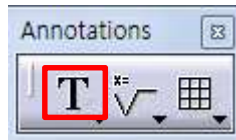


CONTENTS

- ✓ 시작하기
- ✓ Views
- ✓ Dimensions
- ✓ **Annotations/Dress-up**

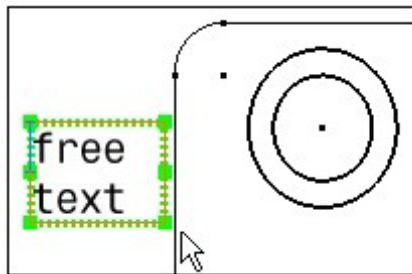
DRAFTING TOOLS

Annotations



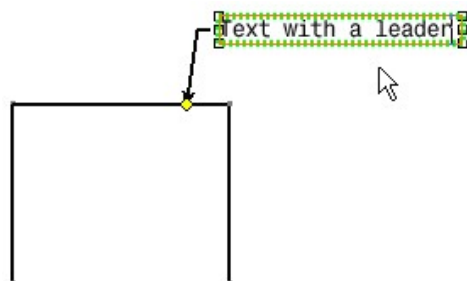
Text T

- 도면에 문자 기입

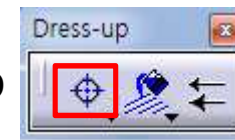


Text with Leader T

- 지시선이 있는 문자 기입

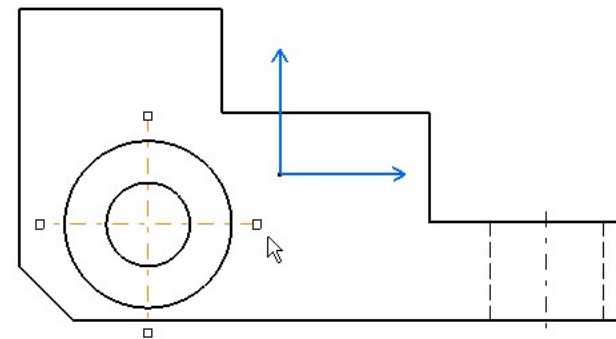


Dress-up



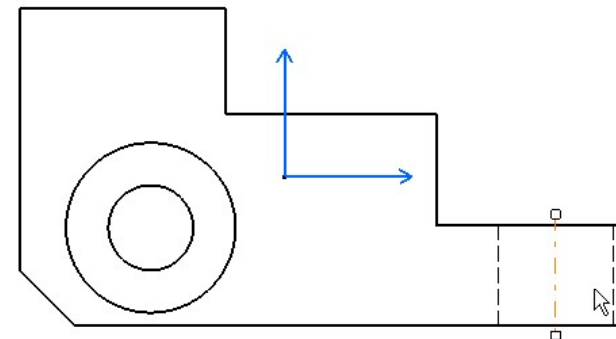
Center Line

- 원의 중심축 생성



Axis Line

- 원기둥의 중심축 생성



실습 과제

다음 도면을 보고 투상도 배치 및 치수 입력 해보기

