김창만 교수와 함께 하는 NX의 달인

목 차

1. Unigraphics(UG)/NX("Next") Software History

- 1969 UNIAPT was released by a software company then called United Computing,
- UNIAPT was one of the world's first end-user CAM products.
- 1973 The company purchased the Automated Drafting and Machining (ADAM) software code from MGS in The code became a foundation for a product called UNI-GRAPHICS, later sold commercially as Unigraphics in 1975.
- 1995 Unigraphics on Microsoft Windows NT.
- 1996 Unigraphics V11.0 was launched. 1997 Unigraphics V13.0 was launched.
- 2001 Unigraphics Version 18 was launched.
- 2002 "Next Generation" version of Unigraphics and I-deas, called NX.
- 2007 UG NX5 was launched. Introduction of Synchronous Technology in NX 5. UGS was purchased by Siemens AG in May 2007, and was renamed Siemens PLM Software.
- 2009 NX6(ST:Synchronous Technology), NX7 was launched by SIEMENS PLM Softwares.
- 2010 NX7.5-launched in mid 2010. 2011 NX8 is due.

2. NX의 중요 기본 핵심어(Keyword)

NX Modeling(CAD) 환경 Keyword

★ Associative

★ Inherit(Legacy)

★ Timestamp Order(Fix at Current Timestamp)

★ Reference(From)

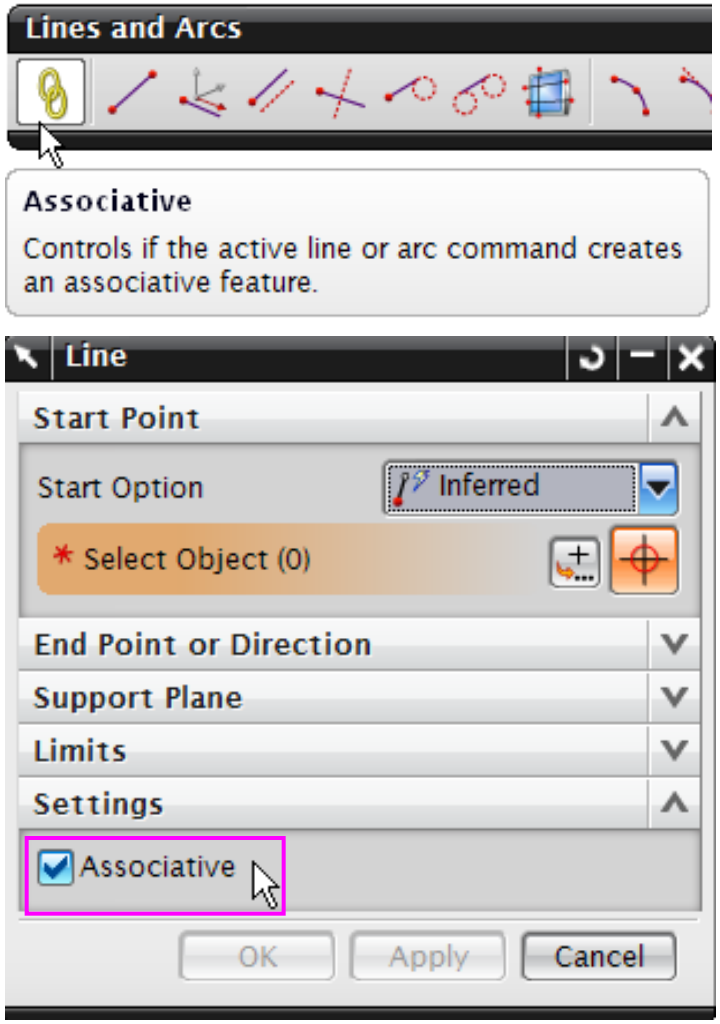
★ Destination(To)

★ By Parameter

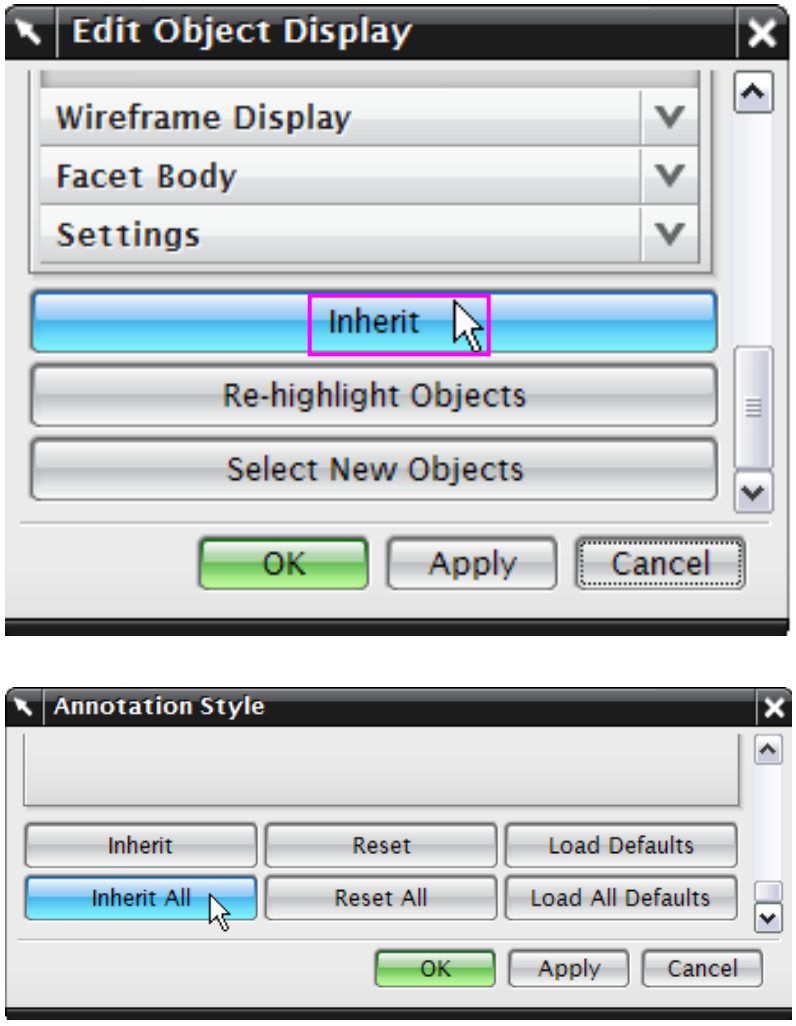
★ As Original

★ Orient Coordinate System/ Datum CSYS/ Point_Plane_Vector Dialog

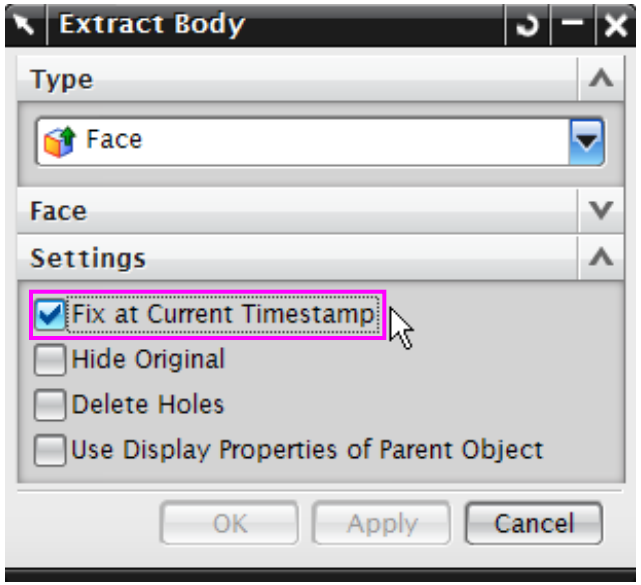
Associative

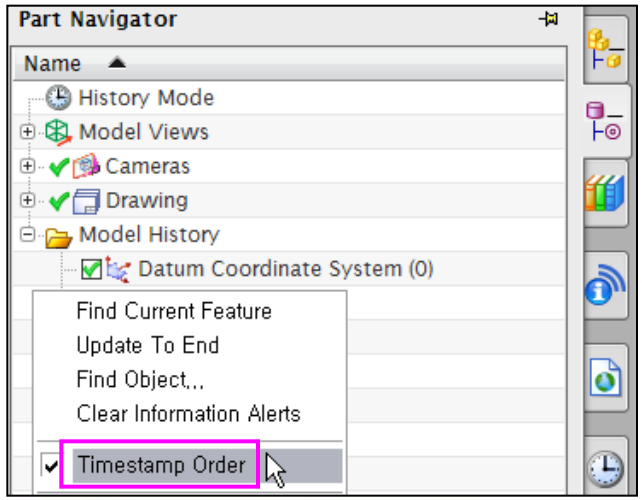


Inherit(Regacy)

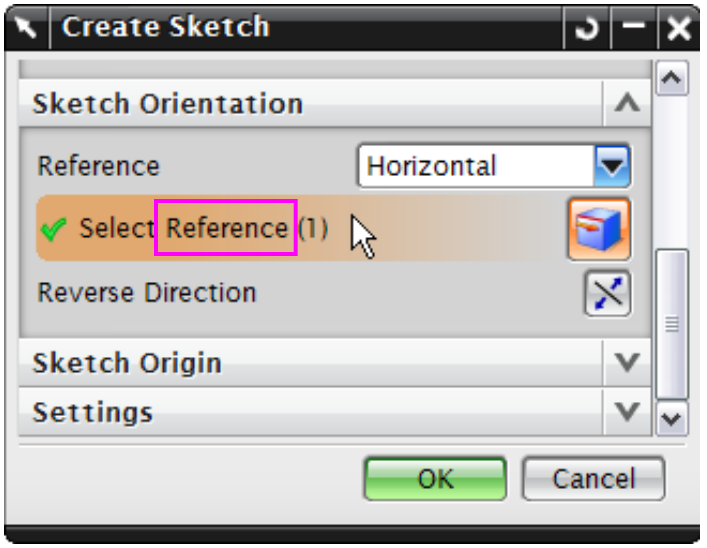


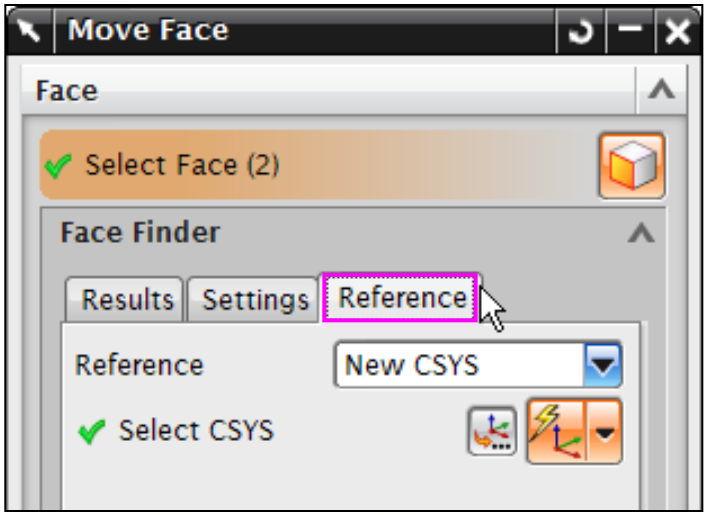
Timestamp Order(Fix at Timestamp)



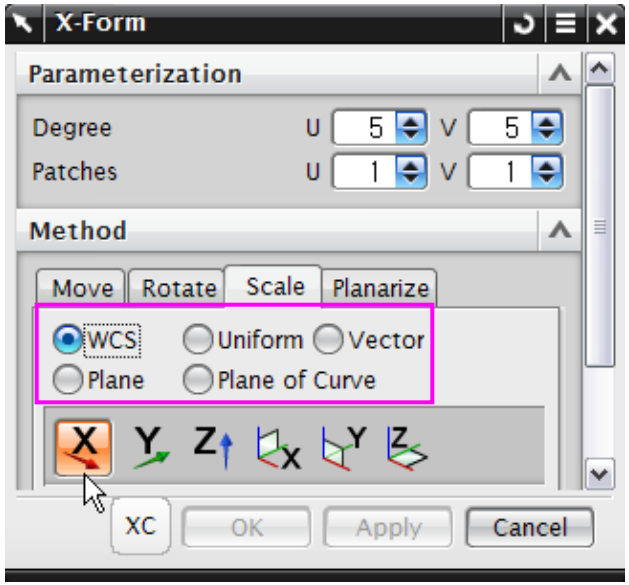


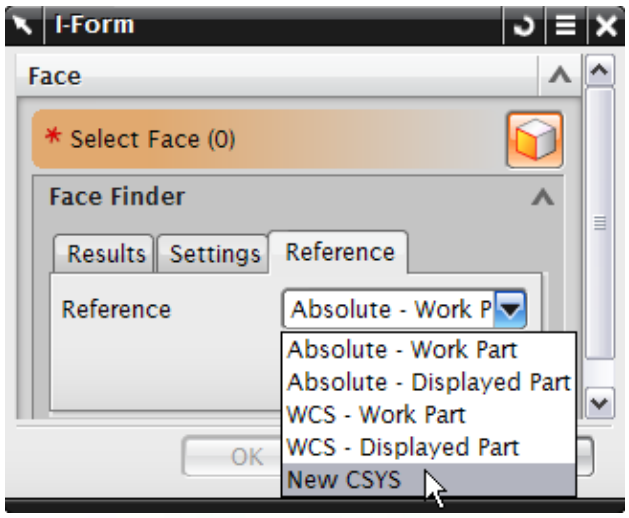
Reference

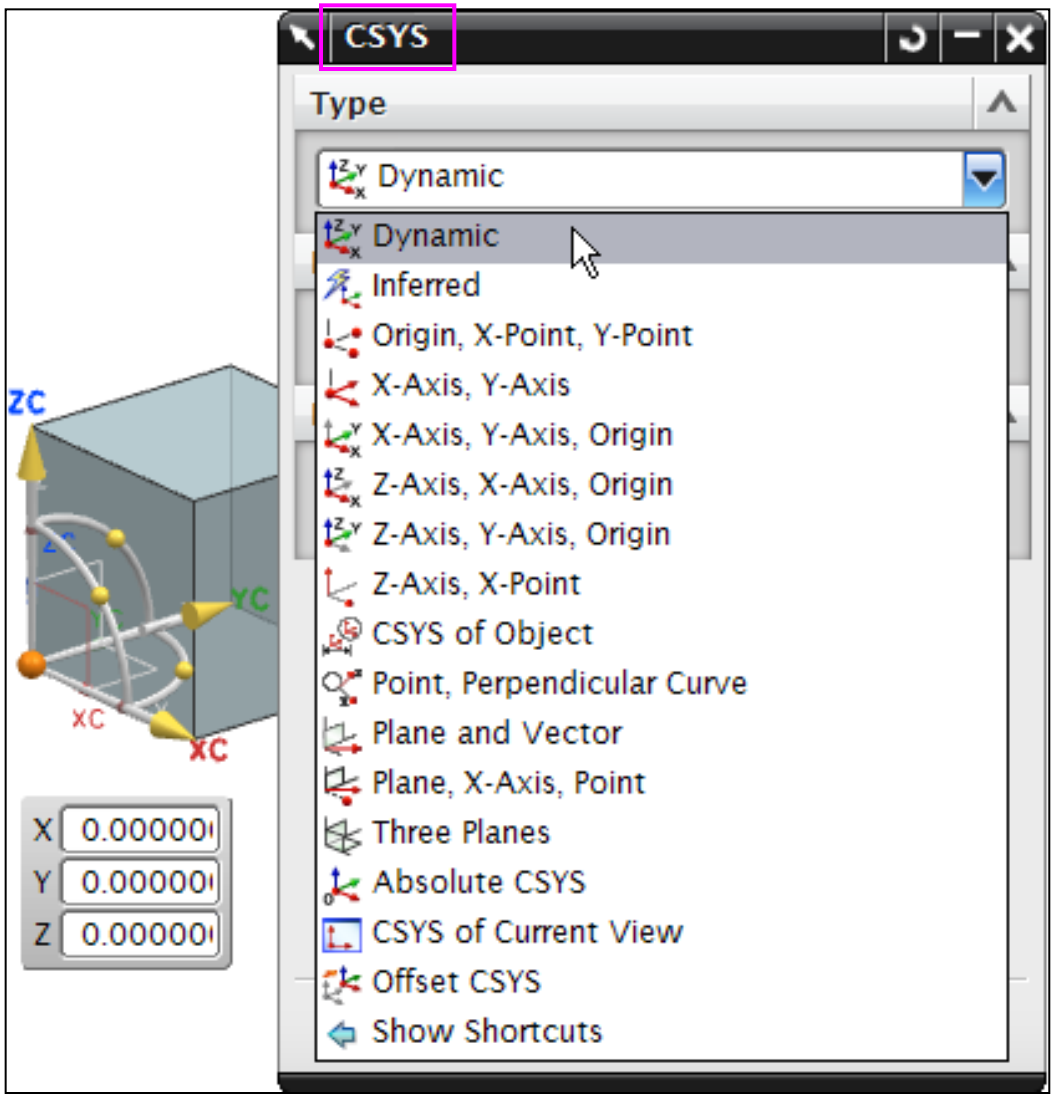




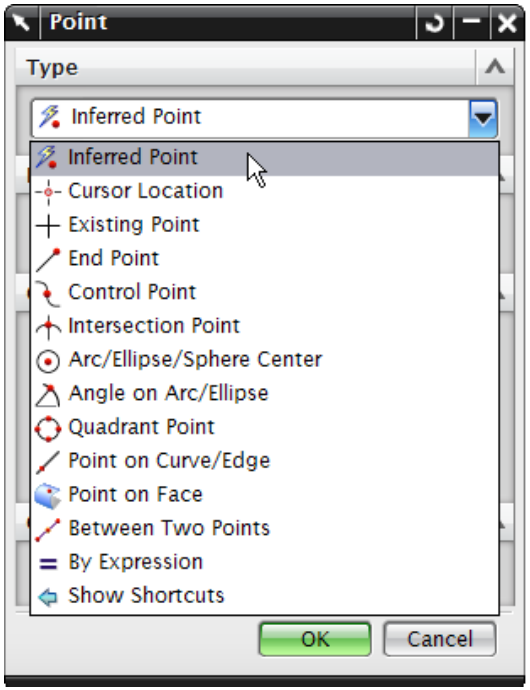
Orient Coordinate System(CSYS): ACS/WCS/MCS/RCS/Datum CSYS



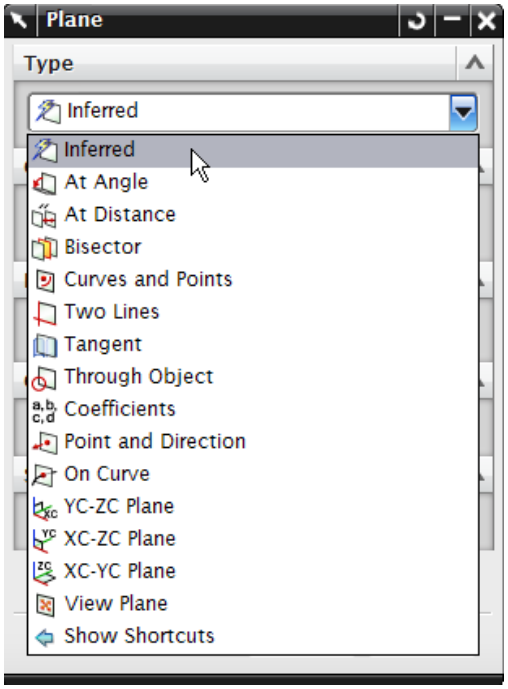




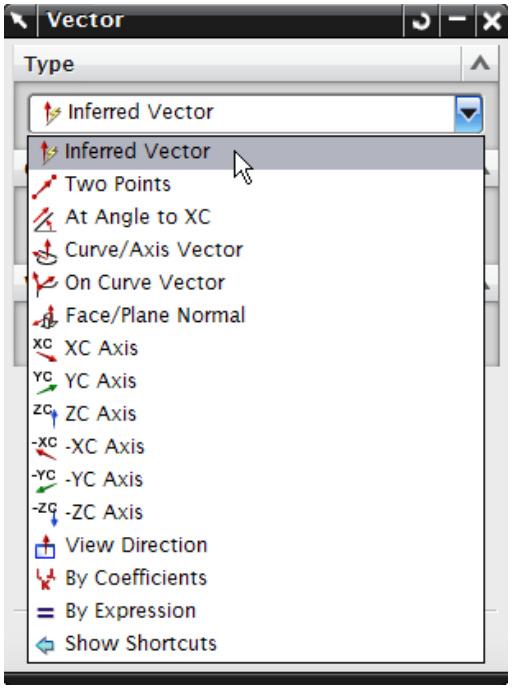
Point Dialog



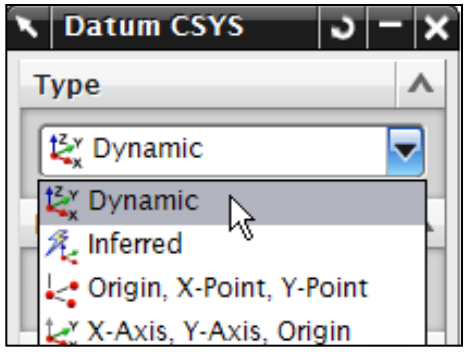
Plane Dialog



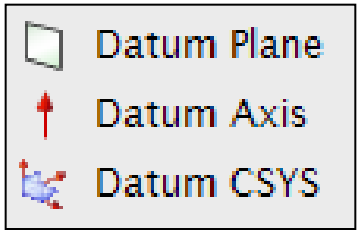
Vector Dialog



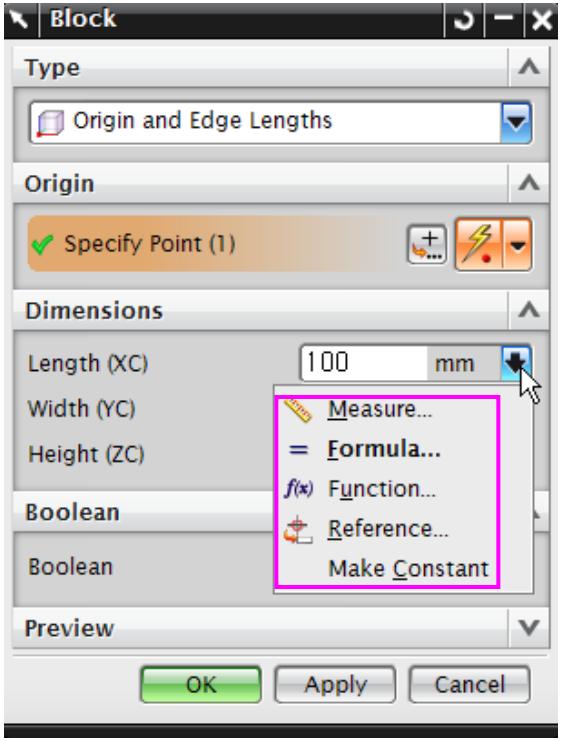
Datum CSYS

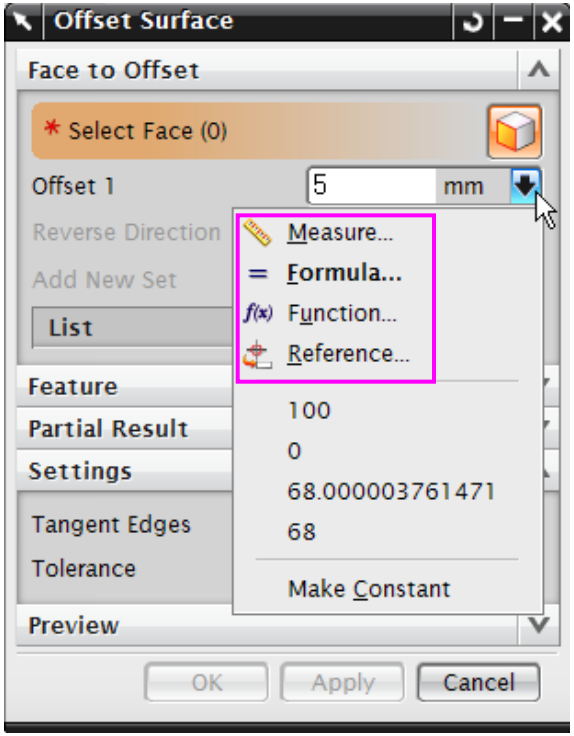


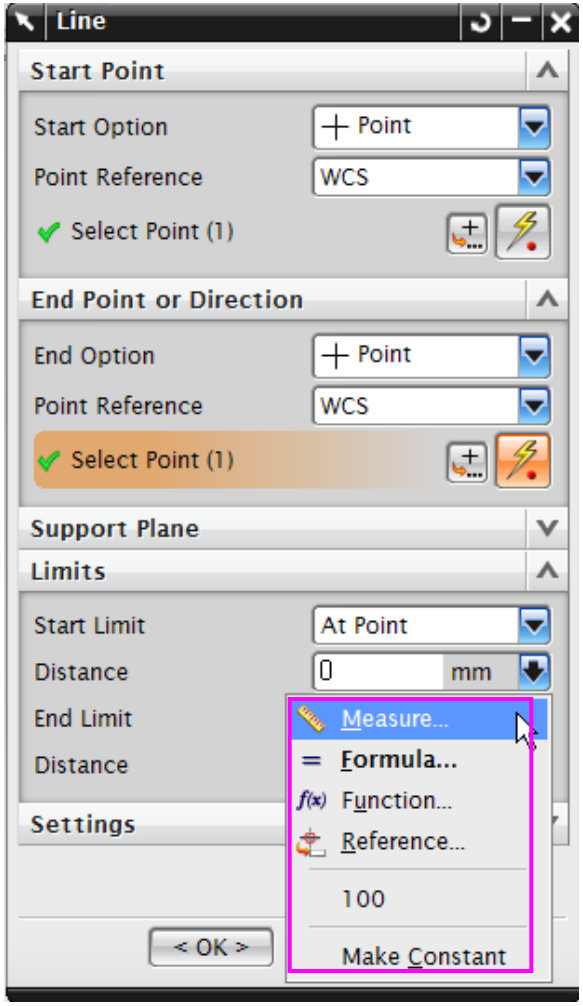
Datum Plane/Axis/CSYS



Objects(Curve, Sheet Body, Solid Body), Section, "Make...", "Work..."
Measure, Formula, Function, Reference, Make Current Feature, Reorder, etc.



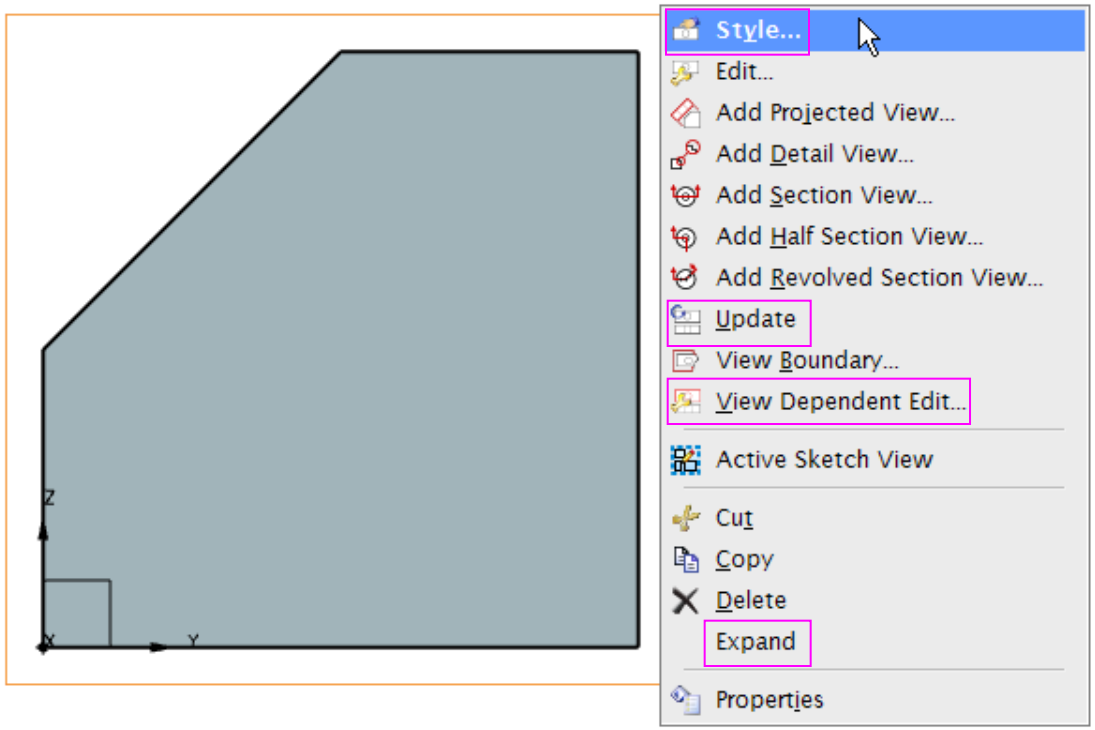




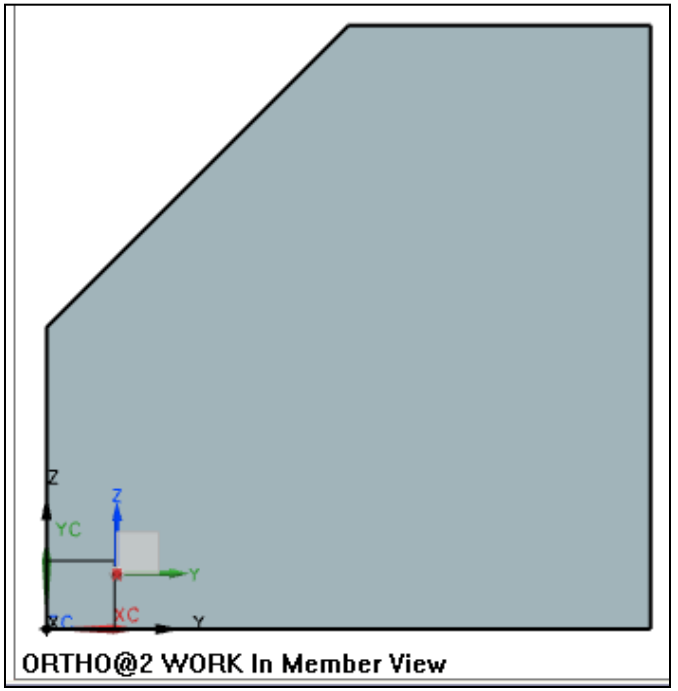
NX Modeling(Drafting) 환경의 중요 기본 핵심어(Key Word)

★ View의 Border 선택:
View Style, Update, View Dependent Edit, Expand(Member View)

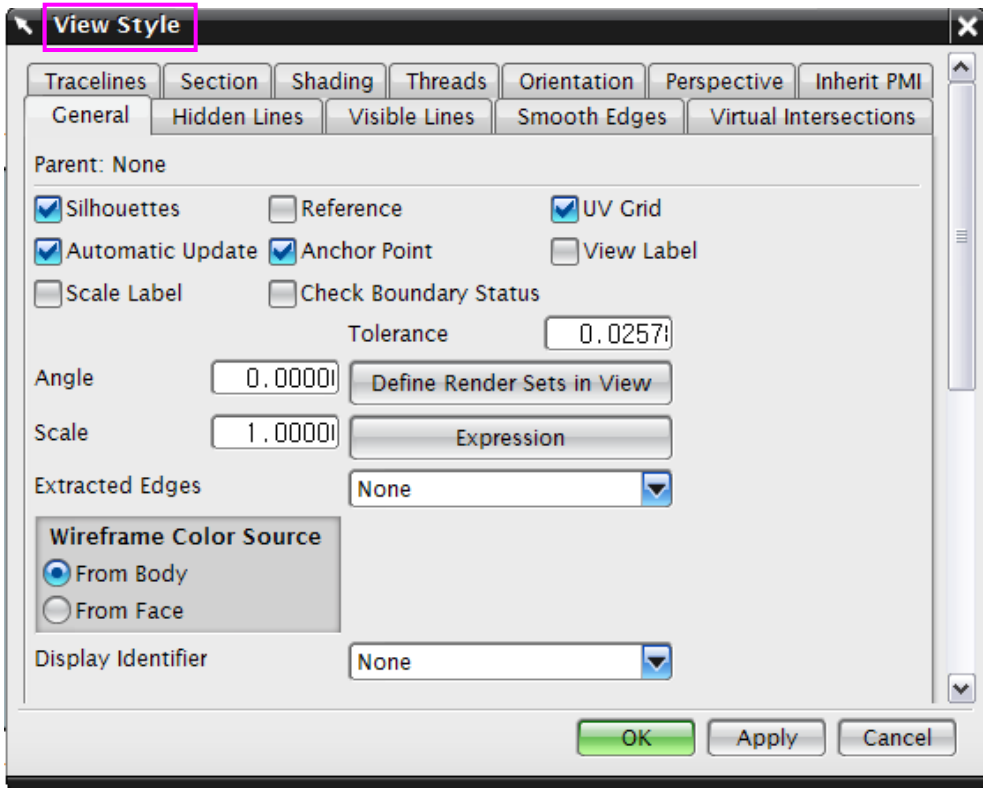
Border 선택 후 MB3



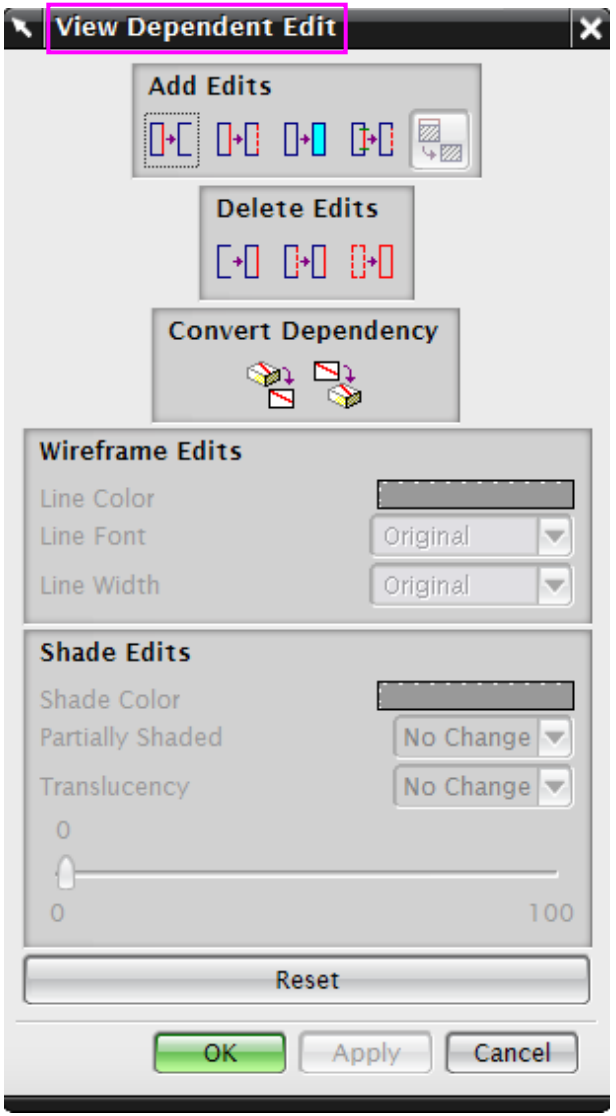
Expand(Member View)



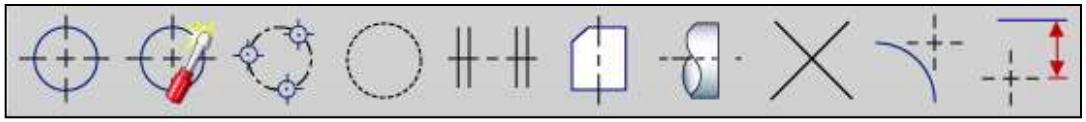
View Style



View Dependent Edit



Annotation Symbol



NX Modeling(Sketch) 환경의 작업 순서

★ Sketch 작업 중 구속여부/개수 확인을 위해 Curve를 Drag하여 DOF 확인

★ Circle/Arc의 중심을 X Axis 또는 Y Axis에 Point on Curve
/축의 원점에 Coincident 형상 구속조건(Constraint Condition) 부여

★ X Axis 또는 Y Axis에 가장 낮은 거리부터 먼저 치수 구속조건 부여

★ X Axis 또는 Y Axis에 좌,우 거리에 치수 구속조건 부여

★ 좌우, 상하 대칭 형상은 가급적 Mirror 기능을 사용

★ 형상 구속조건 먼저 부여 후 치수 구속조건 부여 순으로 작업

★ 필히 Sketch 작업 후 Analysis에서 구속이 유지되는지 치수정보 확인

★ Drafting 치수 중복 작업을 줄이려면 치수 구속을 많이 부여(Feature Parameters)

NX Modeling(Assembly) 환경의 작업 순서

★ Create Component Array/Mirror Assembly/Create Interpart Reference
WAVE Geometry Linker/Promote(Body) 등의 Assembly 작업을 최대한 활용

★ Assembly 작업 중 구속여부/개수 확인을 위해 Show DOF )확인

★ Assembly Navigator의 조립 구속(Assembly Constraints) 오류 확인

★ 조립 부품 간의 시각적 충돌 검사(Move Component-Collision Detection) 확인

★ Assembly Sequence에 의한 부품 간의 동적 분해/조립 및 충돌 검사 확인

★ 조립 부품 간의 끼워맞춤에 대한 Analysis의 조립 간극 체크 확인
(Analysis - Assembly Clearance - Simple Clearance Check)

★ 조립 부품 간의 Analysis의 단순 간섭 체크(Simple Interference) 확인

NX Manufacturing(CAM) 환경의 중요 기본 핵심어(Keyword)

★ Geometry, Part, Blank, Check, Cut Area, Trim Boundaries

★ Analysis(NC Assitant)/Tool, Tool Axis Setting/MCS, RCS

★ Cutting Parameters
(Stock, Intol/Outol, Cut Levels, IPW, Level/Depth First, Inward/Outward, On Part/Plane, Tool Position(On, Tanto, Contact), Climb/Conventional Cut)

★ Feed and Speeds(Optimize Feed Rate, Rapid, Approach, Engage, Cut, Step Over, Traversal, Retract, Departure, Spindle Speed)

★ Non Cutting Moves(Engage, Retract, Transfer/Rapid, Avoidance)

★ Generate/Parallel Generate Tool Path

★ 2D/3D Verify(Gouge Check, Excess, Collision Check, Machine Simulation)

★ Post Process, Shop Documentation, Batch Process, Post Builder

3. NX의 중요 Icon(쪽그림/그림문자)과 Icon 형상정보는 답(쏩)

★ 평소 입으로 명령어를 소리 내 발음하며 클릭하는 습관을 가져야 아이콘이 오래 기억된다.

★ 초급 입문자 일수록 모든 아이콘 명령어를 화면에 나열해두고 눈으로 늘 익히길 바란다.

★ 프린터로 화면을 출력해두고 자신이 알고 있는 아이콘 명령어를 영어로 혼자 발음하며 기억해본다.

★ 아이콘 이미지가 자신이 해결할 문제의 정답 그림이라고 생각하고 아이콘 이미지처럼 따라 모델링 해본다

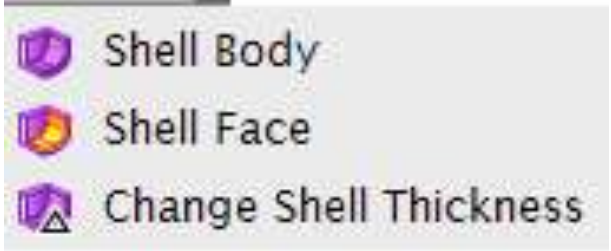
★ 이후에는 아이콘의 명령어가 어떤 기능을 하는지 머릿속에 함께 떠올려보며 Full Menu Bar를 속독해 본다.

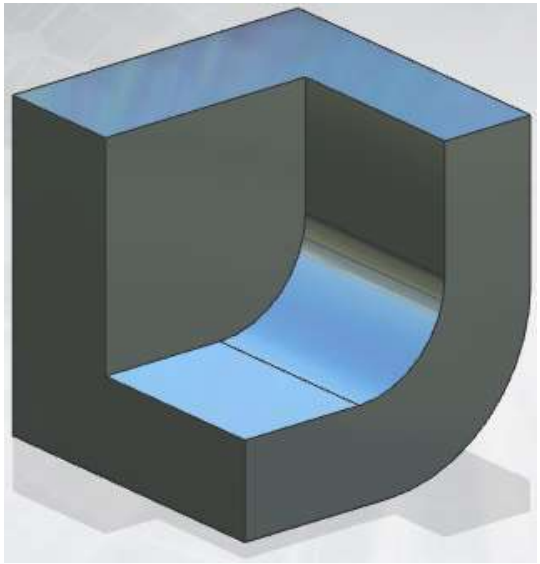
Ctrl+Shift+MB1 Ctrl+Shift+MB2 Ctrl+Shift+MB3

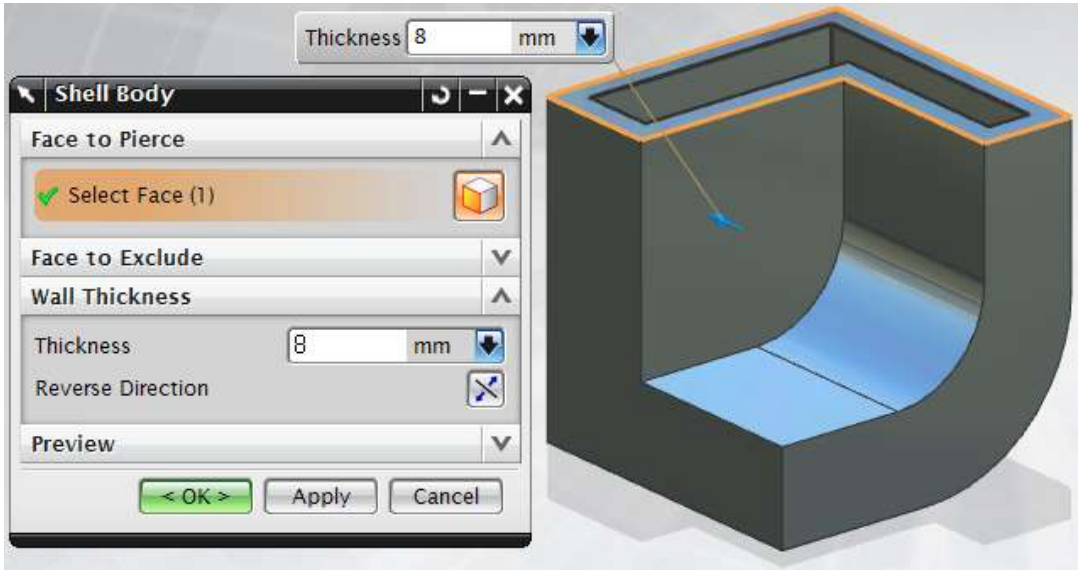
Icon 형상정보는 (사용자 자신이 설계하려는 Part의) 답(쏩)!

★ 아이콘 이미지가 자신이 해결해야 할 문제의 정답 그림이라고 생각하고 아이콘 이미지처럼 따라 모델링 해본다

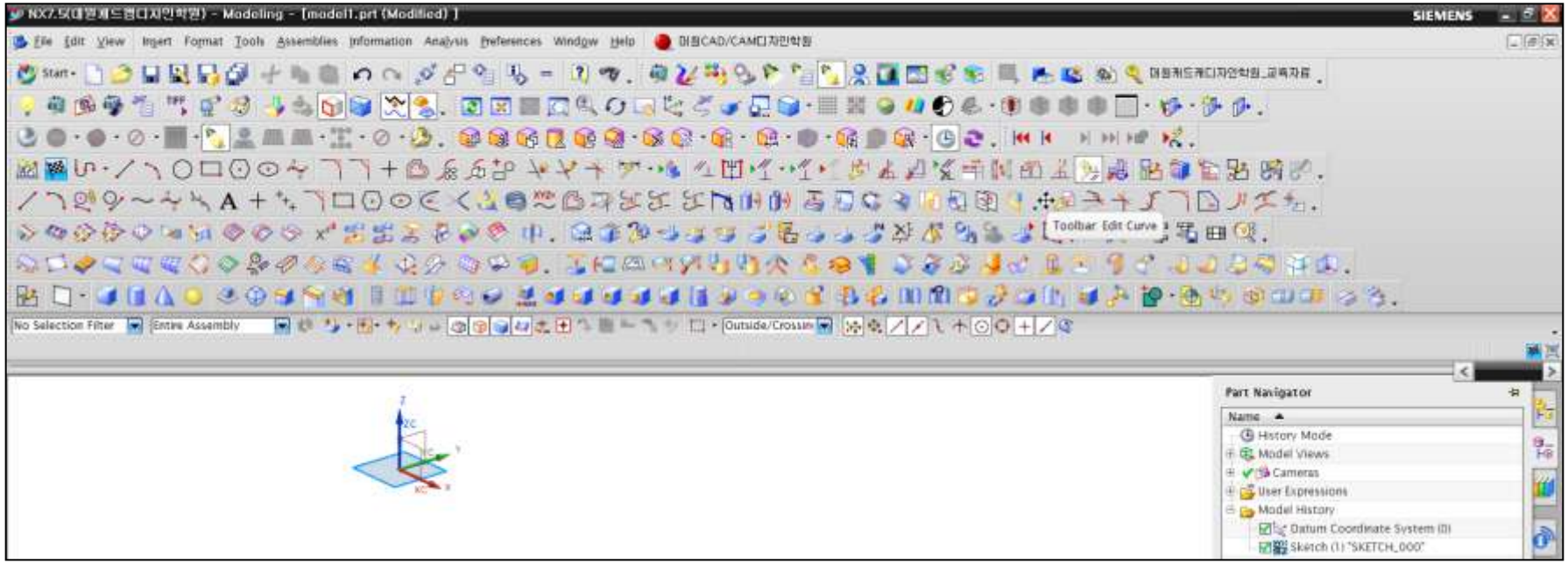
예) Shell Body



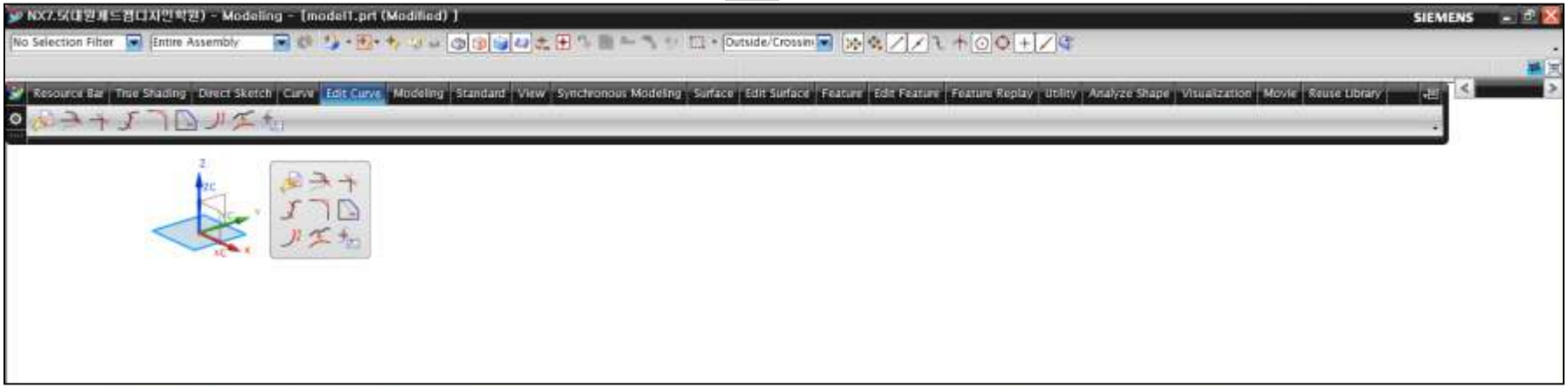




NX Modeling 기본 환경의 작업창 및 Icon Toolbar 배열(예)

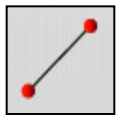


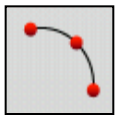
NX Modeling Full Screen(Alt + Enter: ▣) 환경의 작업창 및 Icon Toolbar 배열(예)

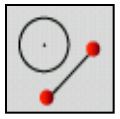


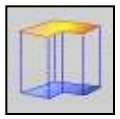
NX 기능을 배우려 할 때 Icon의 순서(1)를 배열 한다면 ?


(Sketch)


(Line)


(Arc/Circle)


(Basic Curve)

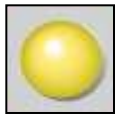

(Extrude)


(Revolve)

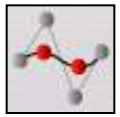

(Block)


(Cylinder)


(Cone)


(Sphere)


(Move Object)

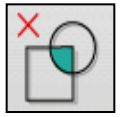

(Studio Spline)


(Trim Body)


(Split Body)


(Unite)


(Subtract)

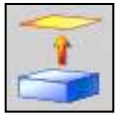

(Intersection)

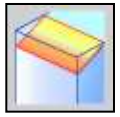

(Edge Blend)


(Face Blend)


(Styled Blend)

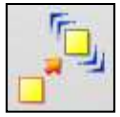

(Extract Curve)


(Extract Face)


(Chamfer)


(Draft)

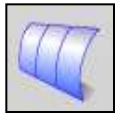

(Draft Body)


(Instance Geometry)

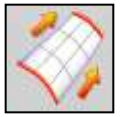

(Sew)


(Unsew)


(Ruled)


(Through Curve)


(Through Curve Mesh)


(Swept)


(Styled Sweep)


(Shell)

NX 기능을 배우려 할 때 Icon의 순서(2)를 배열 한다면 ?


(Move Face)


(Pull Face)


(Offset Region)


(Resize Face)


(Replace Face)


(Resize Blend)


(Delete Face)


(Pattern Face)


(Cut Face)


(Paste Face)


(Mirror Face)


(Linear Dimension)


(Angular Dimension)

(Radial Dimension)


(Radial Dimension)

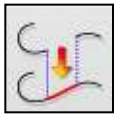

(Replace Blend)


(Group Face)

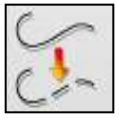

(Optimize Face)


(Resize Chamfer)

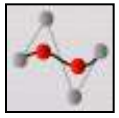

(Label Chamfer)


(Bridge Curve)

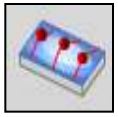

(Join curve)

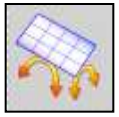

(Simplify Curve)


(Curve Length)


(Studio Spline)


(X-Form)


(I-Form)


(Swoop)


(Trimmed Sheet)

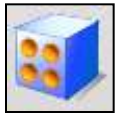

(Trim and Extend)


(Hole)


(Offset Surface)


(Thread)

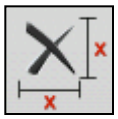

(Combined Projection)

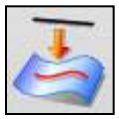

(Instance Feature)

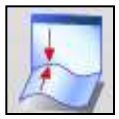
NX 기능을 배우려 할 때 Icon의 순서(3)를 배열 한다면 ?


(Offset Curve)

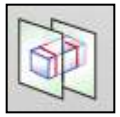

(Wrap/Unwrap Curve)

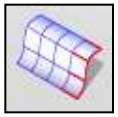

(Remove Parameter)


(Project Curve)


(Combined Projection)


(Intersection Curve)


(Section Curve)


(Extension)

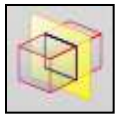

(Law Extension)


(Emboss)


(User Defined Feature)

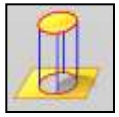

(Mirror Feature)

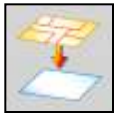

(Mirror Body)


(Emboss Sheet)

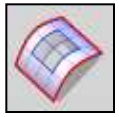

(Bounded Plane)


(Scale Body)


(Patch Openings)


(Quilt)

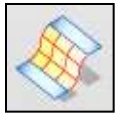

(N-Sided Surface)


(Enlarge)


(Divide Face)


(Join Face)

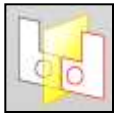

(Thicken)


(Bridge)

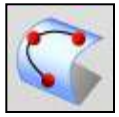

(Text)

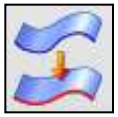

(Tube)


(Slot)


(Mirror Curve)


(Offset Curve in Face)


(Curve on Surface)


(Change Edge)


(Match Edge)


(Edge Symmetry)

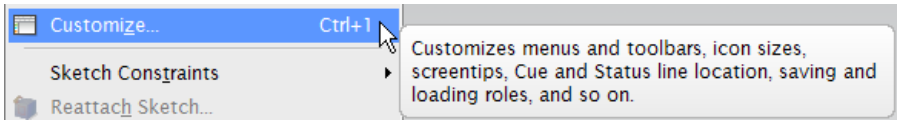

(Point Set)

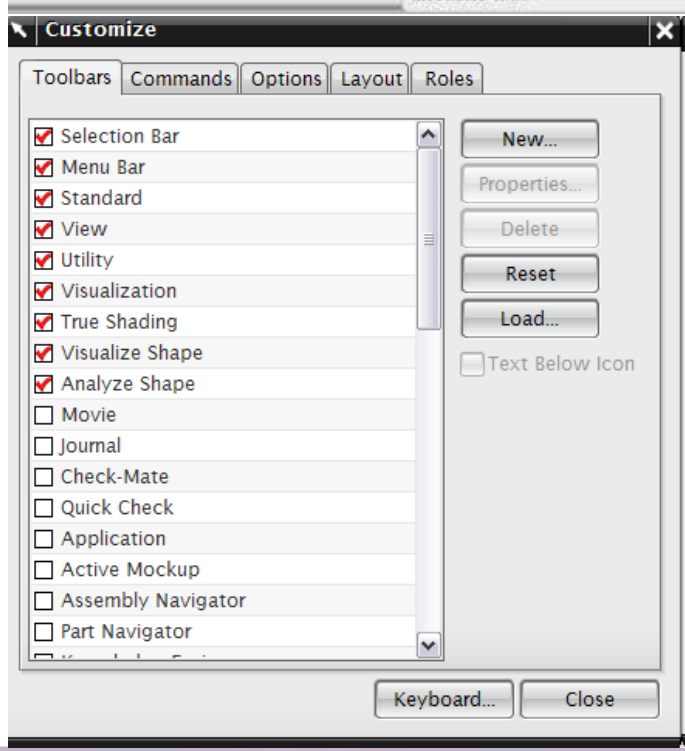

(Midsurface by Face Pairs)

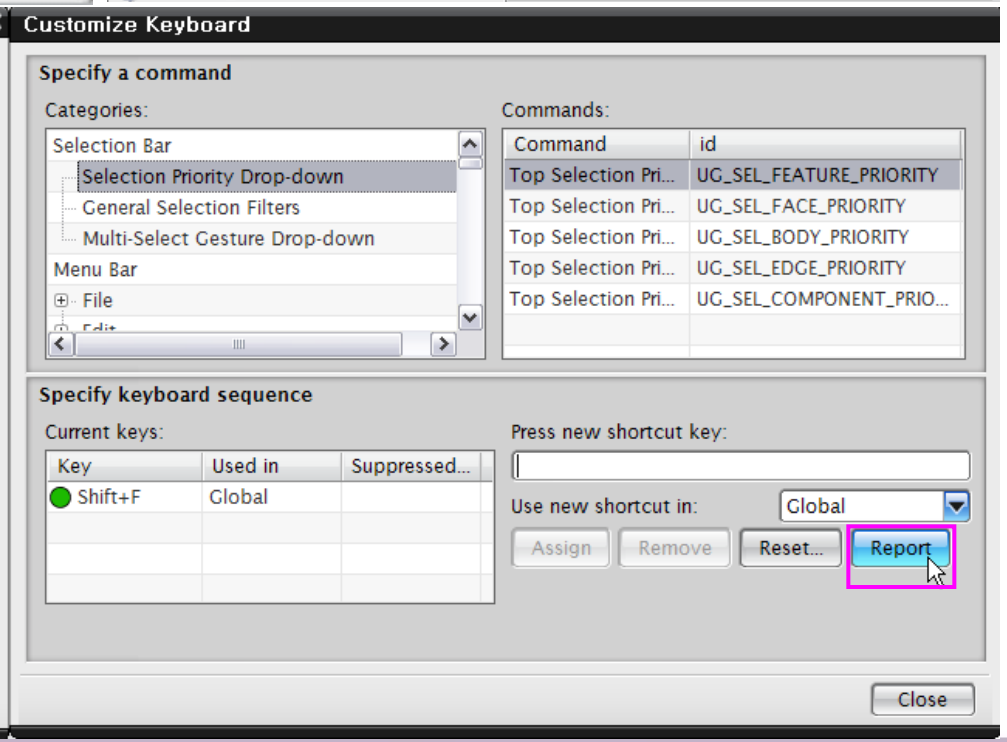
4. NX의 중요 단축키(Shortcut key) 및 Roles/Template 중요성

Information→Custom Menu Bar→Accelerations

Full Menu Bar→Tools→Customize(Ctrl+1)







① File에 관한 단축키

Ctrl+N: 새로만들기, 새 파트 파일(New)

Ctrl+O: 기존의 파일 열기(Open)

Ctrl+S: 즉시(동일 파일 이름) 저장(Save)

Ctrl+Shift+A: 다른 이름으로 저장(Save As)

Alt+F4: 현재 프로그램을 바로 종료 시킴, NX 프로그램 종료(EXIT)

② Application Program 실행에 관한 단축키

Ctrl+M: Modeling 응용 프로그램 실행

A: Assemblies 응용 프로그램 실행

S:: Sketch 응용 프로그램 실행

Ctrl+Shift+D: Drafting 응용 프로그램 실행

Shift+M: Manufacturing 응용 프로그램 실행

③ Edit에 관한 단축키

Ctrl+Z: 하던 작업을 원 상태로 돌려놓기((Undo)/반대 Redo:Ctrl+Y)

Ctrl+X: 원본 파일 전체를 잘라내기(Cut)

Ctrl+C: 원본 파일은 남겨 두고 복사하기(Copy)

Ctrl+D: 파일이나 기타 물체(Object) 삭제하기(Delete)

Ctrl+A: 선택된 파일이나 물체(Object)등의 전체 선택(Select All)

Ctrl+V: 붙이기(Paste)

Ctrl+W: Show and Hide(모두 또는 Group 별로 일괄 숨기기/보이기)

Ctrl+B: Hide(원하는 Object만 숨기기, 숨기기 할 때 마다 Hide 명령어 실행)

Ctrl+Shift+I: Immediate Hide(한번 실행 후 원하는 Object를 즉시 연속해서 숨기기)

Ctrl+Shift+B: Invert Shown and Hidden(Show와 Hide된 Object 반전)

Ctrl+Shift+K: Show(원하는 Object만 다시 보이기)

Ctrl+Shift+U: Show All(숨기기한 Object를 다시 처음처럼 화면상에 모두 보이기)

Ctrl+T: Transform(Scale, Mirror, Array, Point Fit외 기능 Move Object로 변경)

Ctrl+Shift+M: Move Object(시각적으로 보면서 이동 Transform의 대부분 기능 흡수)

Ctrl+J: Object Display

④ View에 관한 기능키/단축키

Ctrl+H: Edit Work Section View

W: 화면상의 WCS(Work Coordinate System) Hide/Display

F5: 작업 중이나, 작업 후 화면의 잔상 제거(Refresh)

Ctrl+F: Object를 화면에 적당히 알맞은 크기로 정렬(Fit)

Ctrl+Shift+Z: Zoom(F6)

Ctrl+I: 선택한 Object의 Information

F4: : Information 창 Hide/Display

Ctrl+R: Rotate(F7)

Ctrl+F8: Restore(복원 : View → Operation → Save As를 해둔 View로 복원)

F3: 작업 중인 Current Dialog Box Hide/Display

F8: Closest Standard View(가장 가까운 표준 뷰 배치 : Top, Left, etc.)

Ctrl+Shift+H: Visualization High Quality Image

: Trimetric Orient View

: Isometric Orient View(등각 방향)

⑤ Format에 관한 단축키

Ctrl+L: Layer Settings

Ctrl+Shift+V: Visible Layers in View

⑥ Tools에 관한 단축키

Ctrl+E: Expressions

Alt+F5: Movie Record

Alt+F7: Movie Stop

Ctrl+Shift+R: Macro Start Record

Ctrl+Shift+P: Macro Playback

Ctrl+Shift+S: Macro Step

⑦ Preferences에 관한 단축키

Ctrl+Shift+J: Object Preferences

Ctrl+Shift+T: Selection Preferences

⑧ Help에 관한 기능키

F1: 기능 설명을 알고 싶을 때 On Context(UGDOC의 Document 지원)

휠(Wheel) 마우스 버튼(Mouse Button : MB) 사용법

(**MB1** : 마우스 왼쪽 버튼, **MB2(Wheel)** : 마우스 가운데 버튼, **MB3** : 마우스 오른쪽 버튼)

MB1: 물체(Object, Entity)/메뉴(Menu)/명령어/커서 위치 선정 등의 기능

MB1(Double Click): 선택된 커서(Cursor)에서 신속히 두 번 클릭하면 Edit with Rollback

MB2+MB3: Shift+MB2 함께 누르는 기능과 동일, Object가 이동되는 Pan 기능

MB2(Wheel) : 손가락으로 휠을 몸쪽으로 당기고 밀면 Zoom In(+)/Zoom Out(-) 됨
Wheel을 잠시 누르면 Set Rotate Point 작동(선택 Point에서 Rotate됨)

MB1+MB2: Ctrl+MB2 함께 누르는 기능과 동일하며 Zoom In/Out 기능이 수행됨

MB3 : 주 기능은 Pop-up 메뉴 지정/옵션(Option) 메뉴 지정/Enter↵ 기능
MB3를 잠시 누르고 있으면 Radial Pop-up Menu가 실행

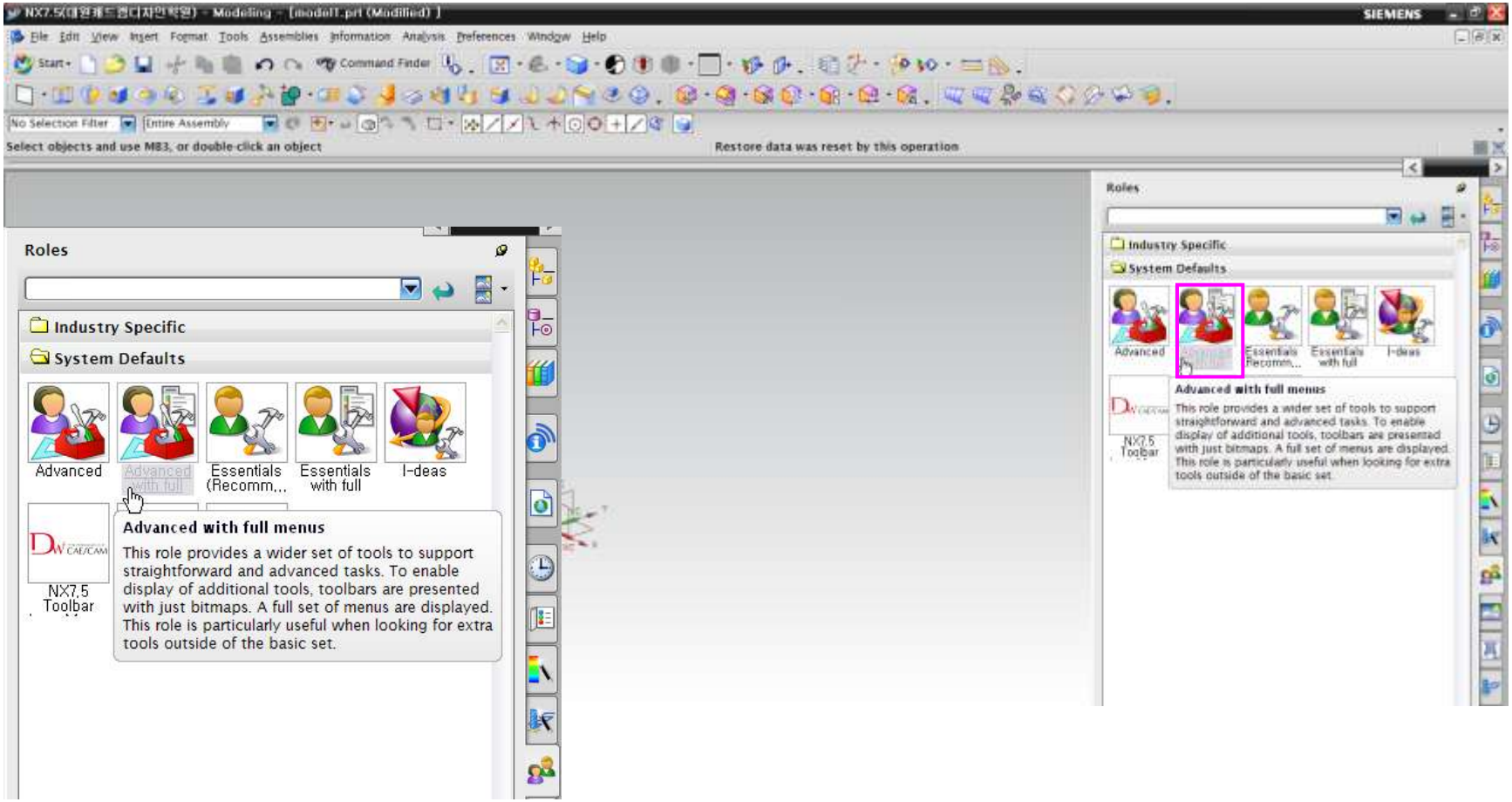
Ctrl+Shift+**MB1** : Application에 등록한 첫 번째의 Radial Pop-up Menu(8개의 Icon) 실행

Ctrl+Shift+**MB2**: Application에 등록한 두 번째의 Radial Pop-up Menu(8개의 Icon) 실행

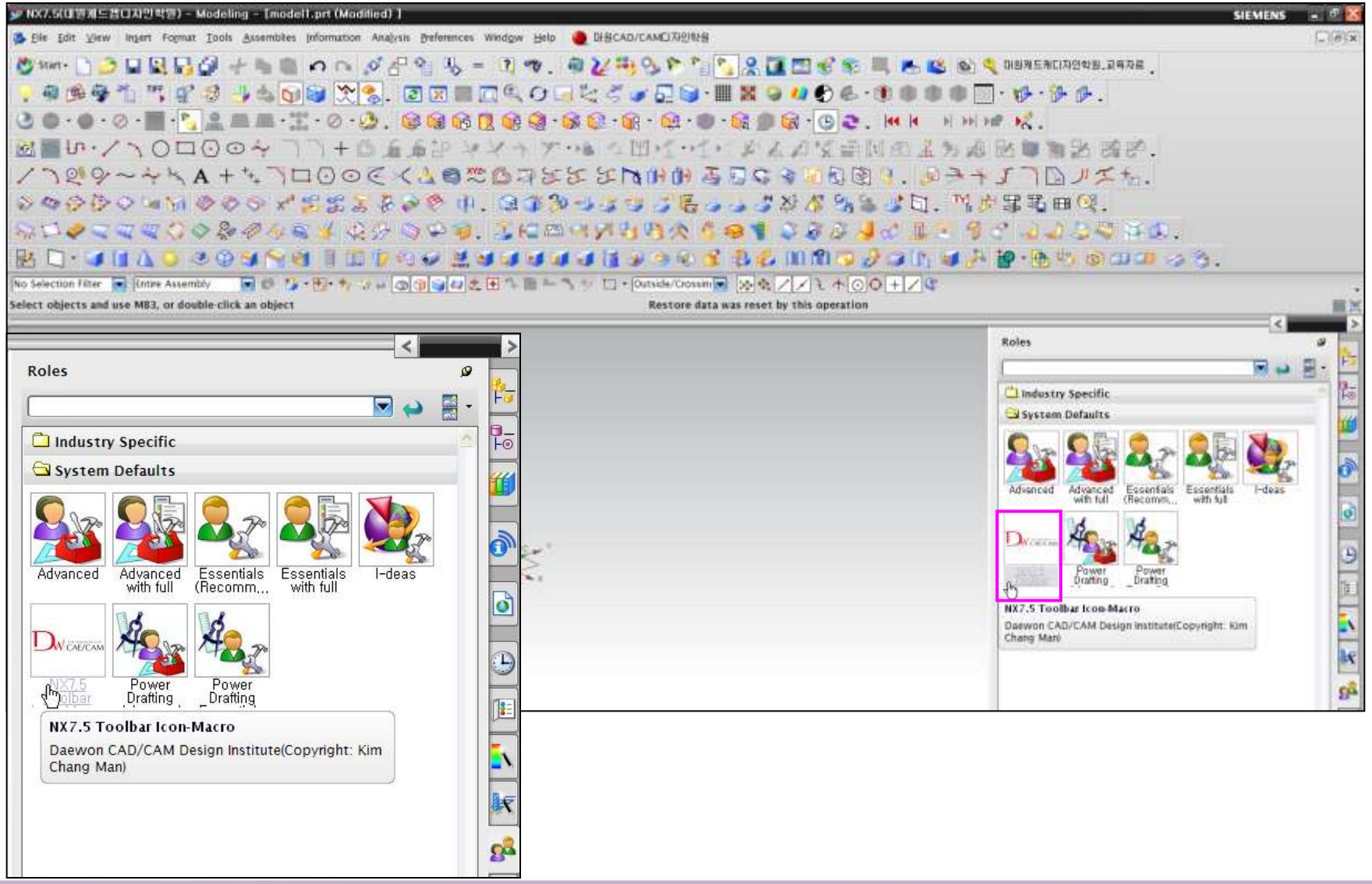
Ctrl+Shift+**MB3**: Application에 등록한 세 번째의 Radial Pop-up Menu(8개의 Icon) 실행

Roles(역할:役割) 중요성

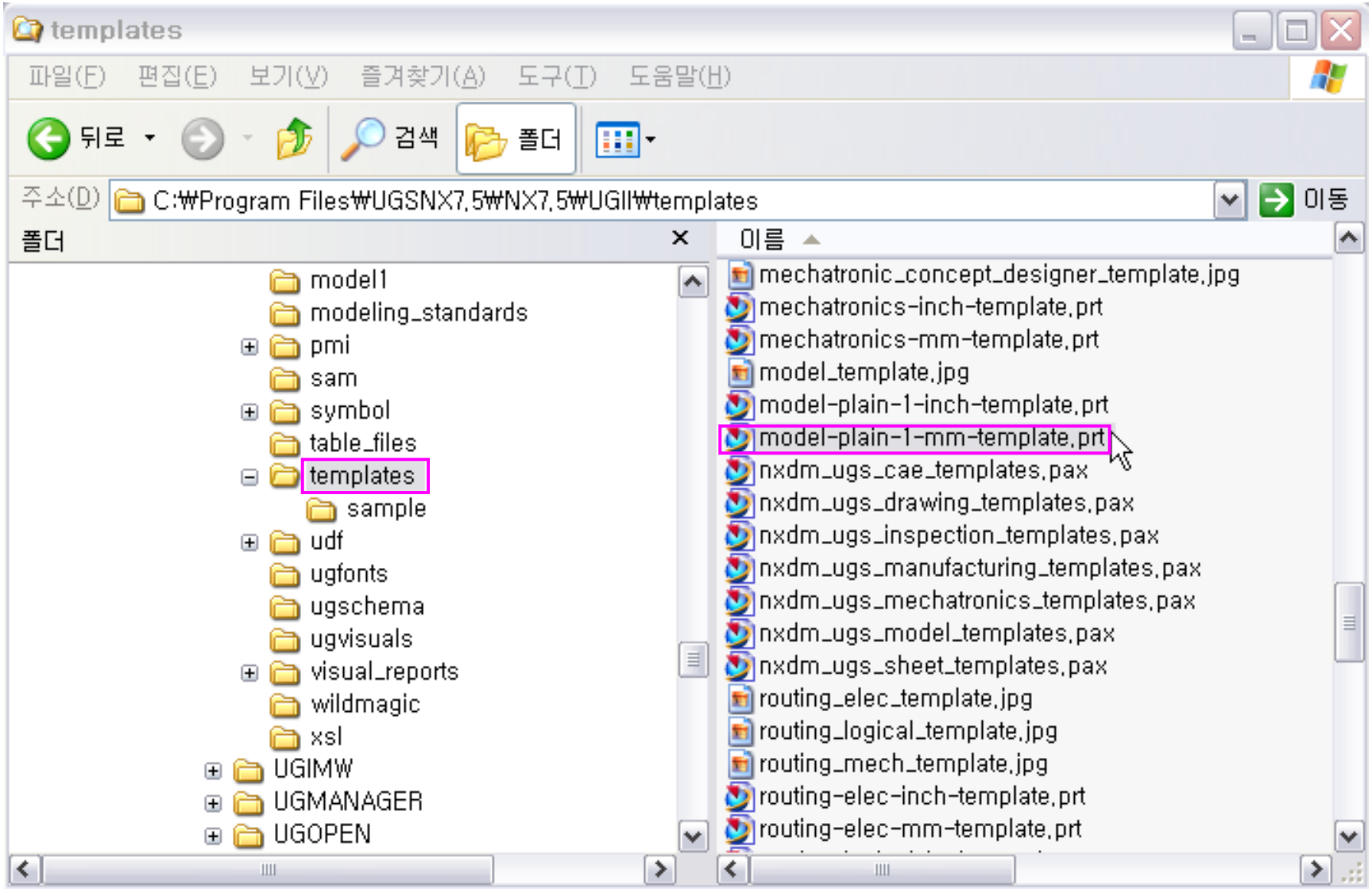
Advanced with full menus Roles 설정 상태



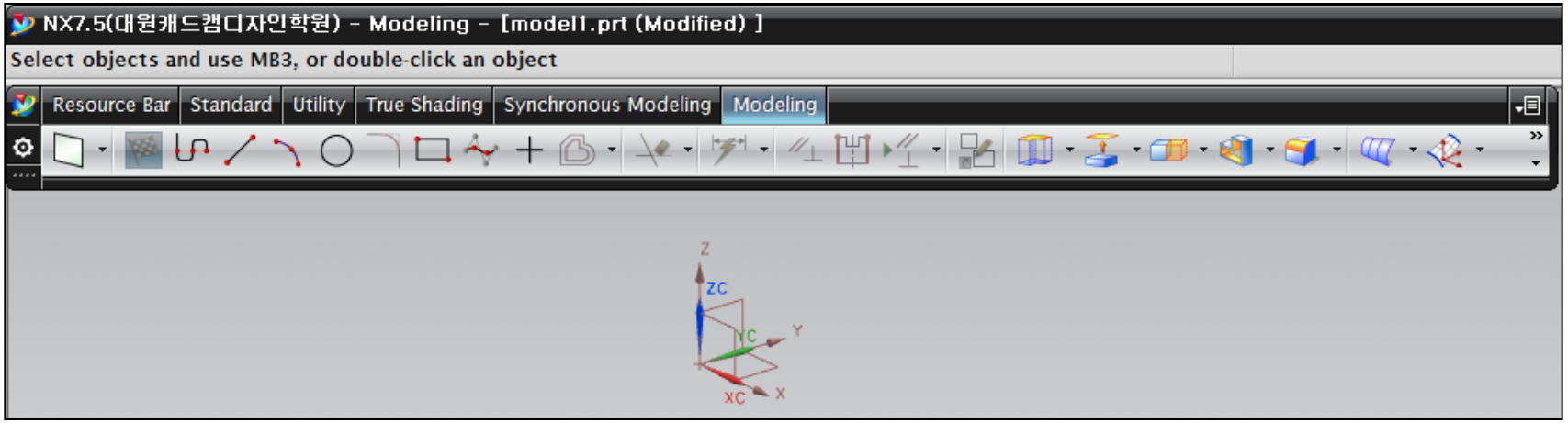
사용자가 설정해둔 Roles 설정 상태



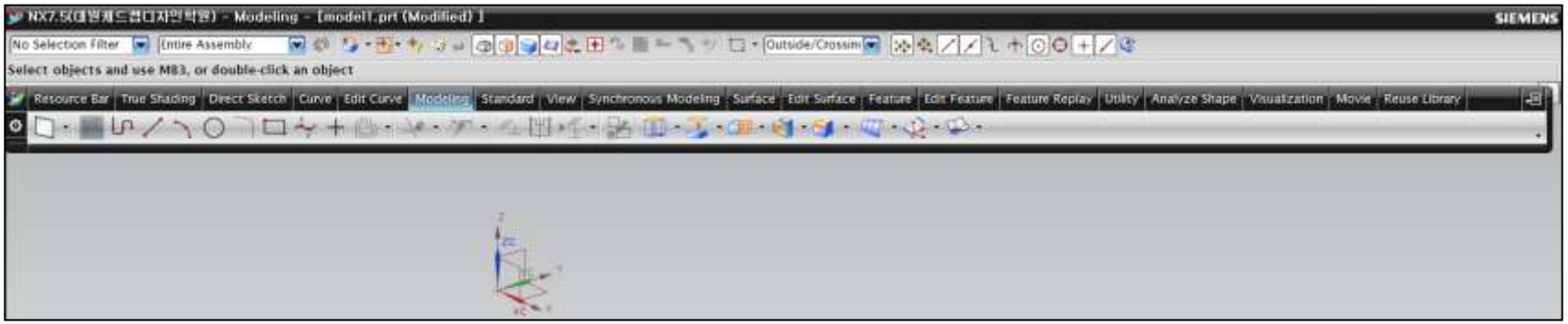
Template(model-plain-1-mm-template.prt) File/Seed File의 Setting



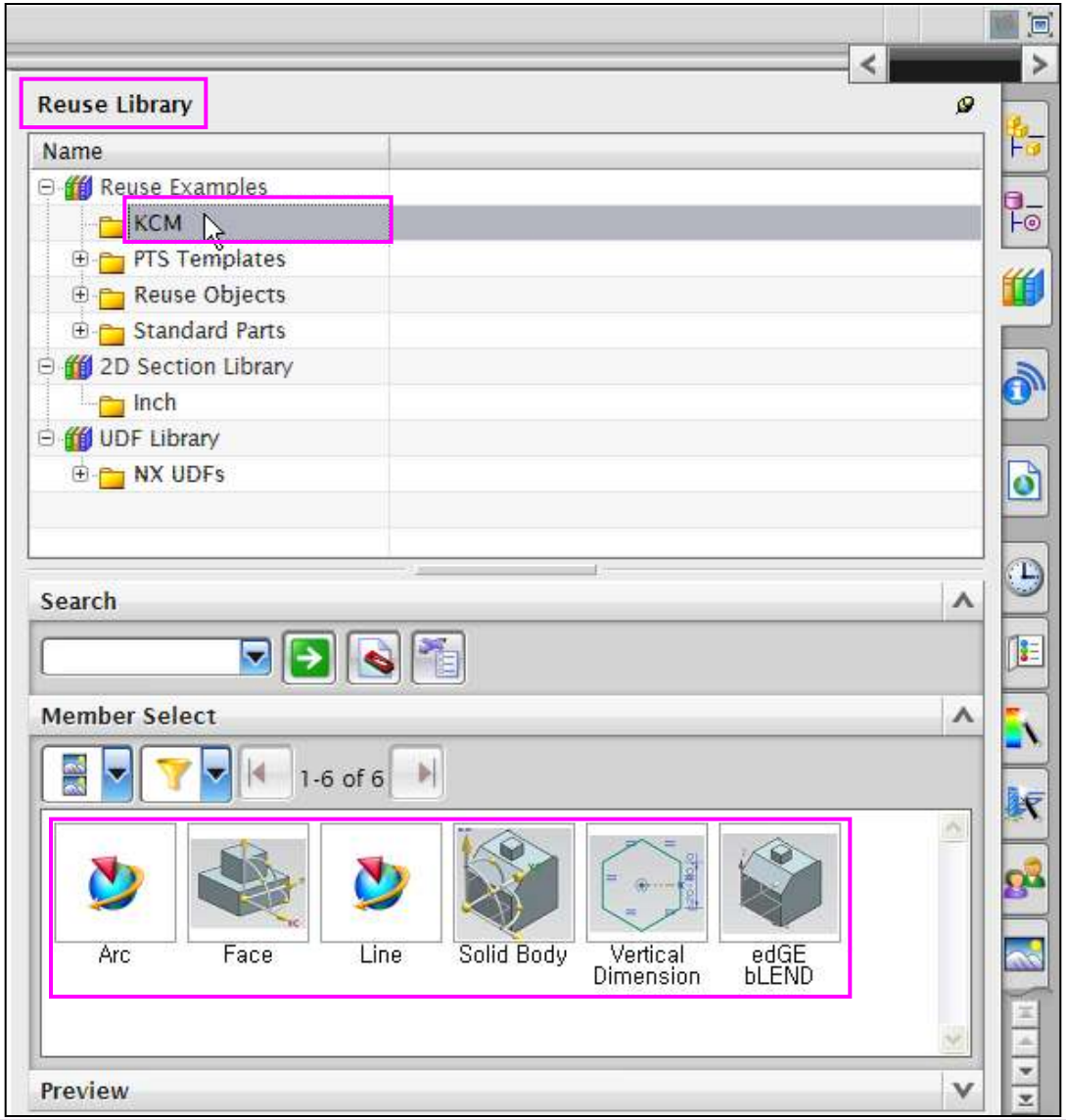
Full Screen Mode(Alt + Enter): Advanced with full menus Roles 설정 상태

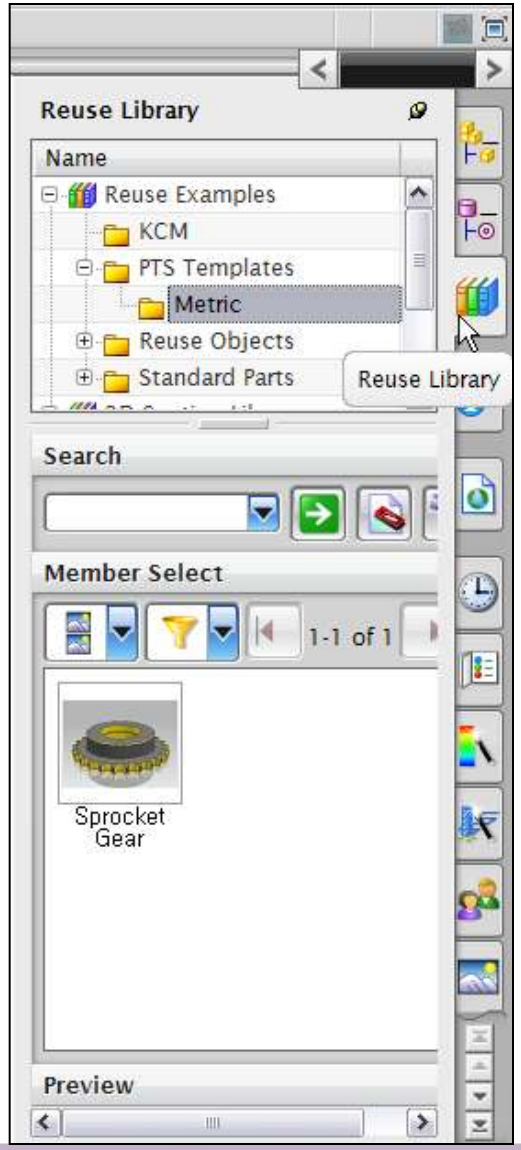


Full Screen Mode(Alt + Enter): 사용자가 설정해둔 Roles 설정 상태

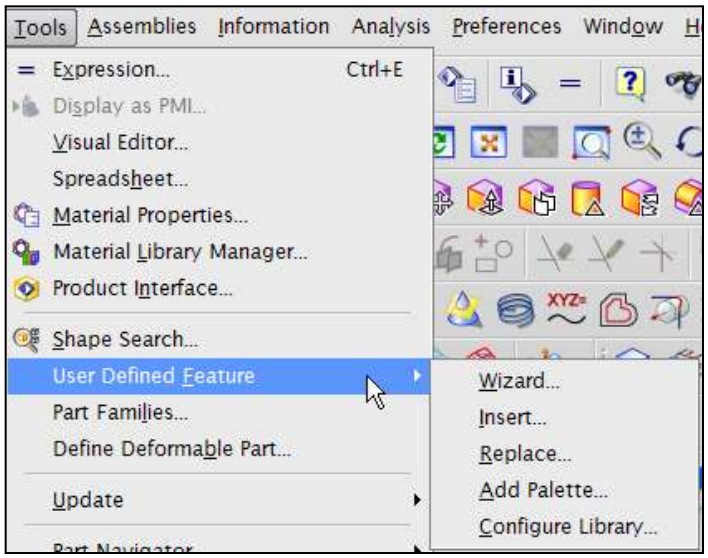


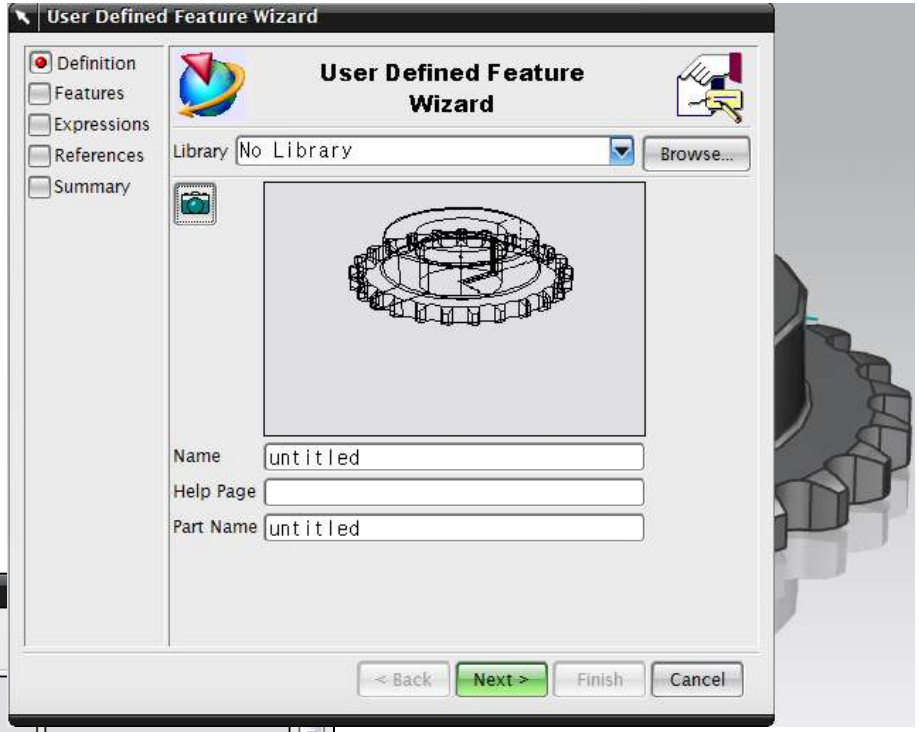
Template(Reuse Library)

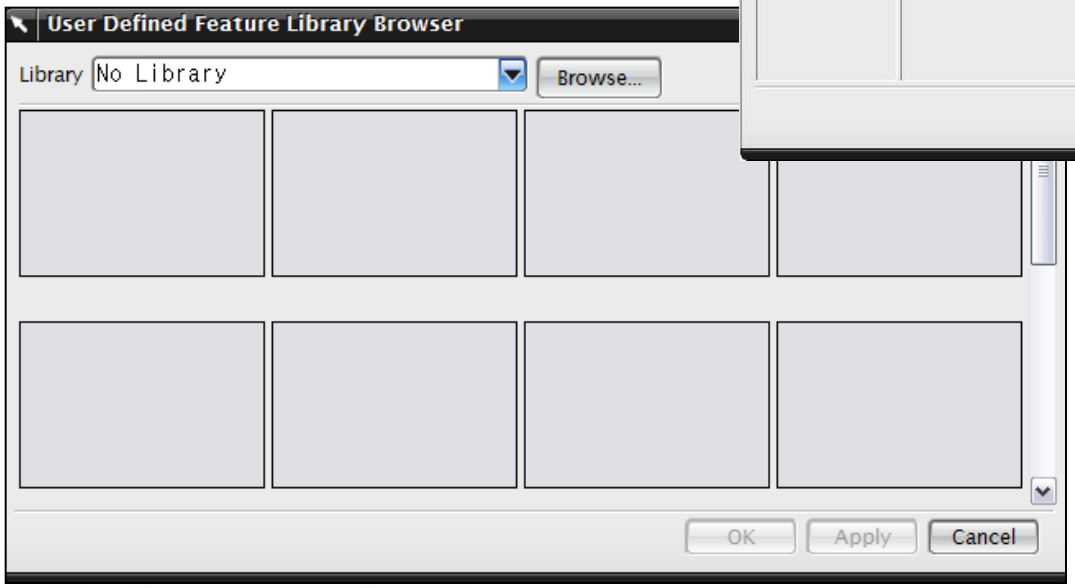




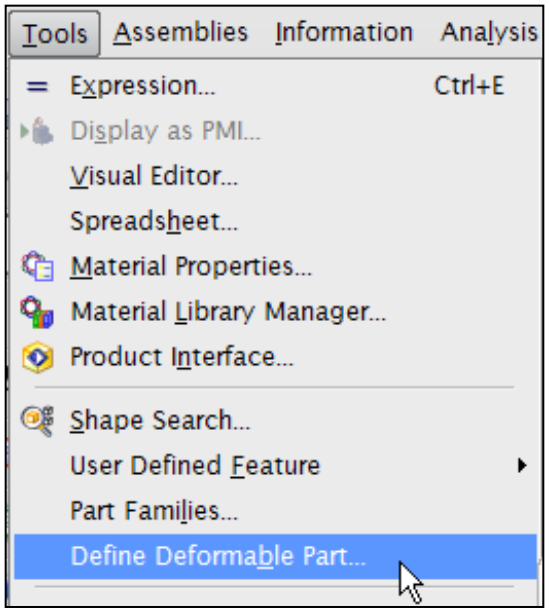
Template(UDF: User Defined Feature)

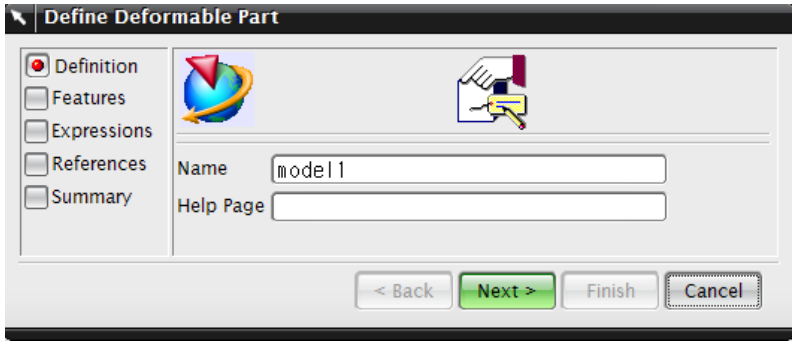


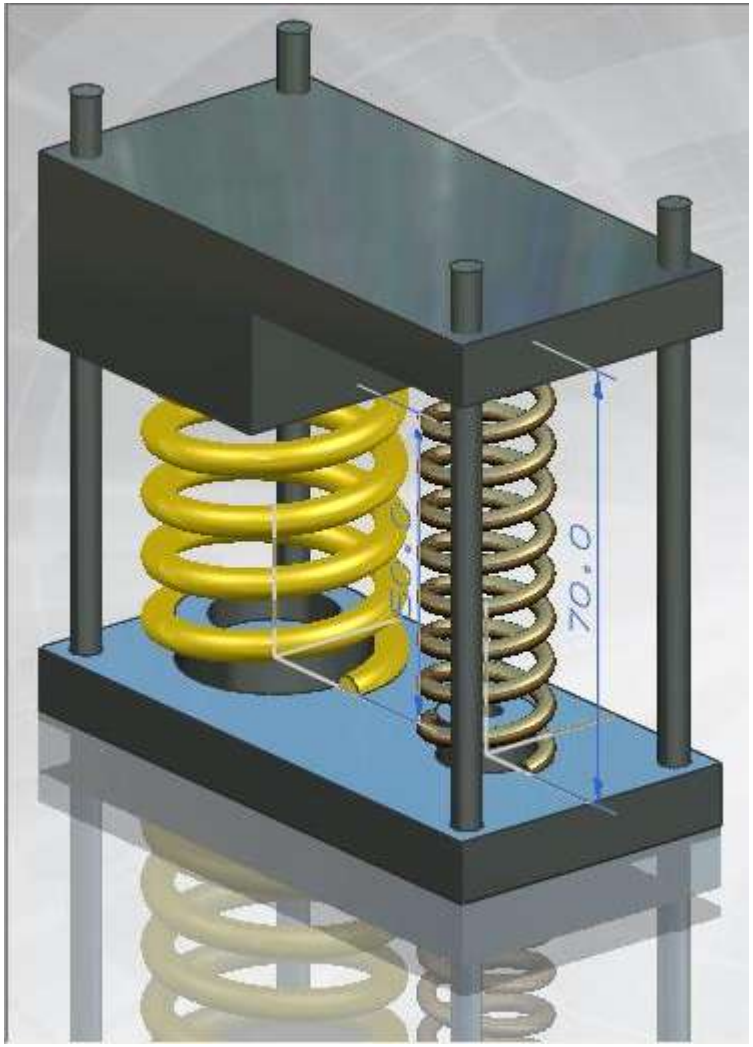




Template(DDP: Define Deformable Part)

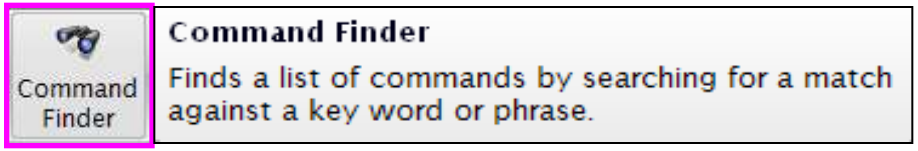






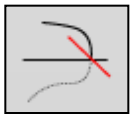
5. 유사 기능의 Icon/ 유사 단어의 명령어는 구구단 암기하듯이 (습관화 한다.)

유사기능의 아이콘끼리 배치해두고 기능을 익히면 하나의 아이콘을 선택하려다
선택 아이콘 좌우의 비슷한 아이콘을 보며 유사기능을 연결 지으며 회상하게 된다.



NX 사용 중에 항상 제어판의 음성, 전자사전과
명령어 찾기는 Active 시켜두는게 달인의 시작!

아이콘 중에 유사 단어와 연관 지어 익히는 것 또한 기억하는데 매우 효과적이다.
예를 들어서 Trim Curve, Trim Corner, Trim Body, Trimmed Sheet 아이콘 처럼
Trim이 포함된 명령어만 검색해서 기능의 차이점을 비교해 보면 더 오래 기억된다.

 (Trim Curve)

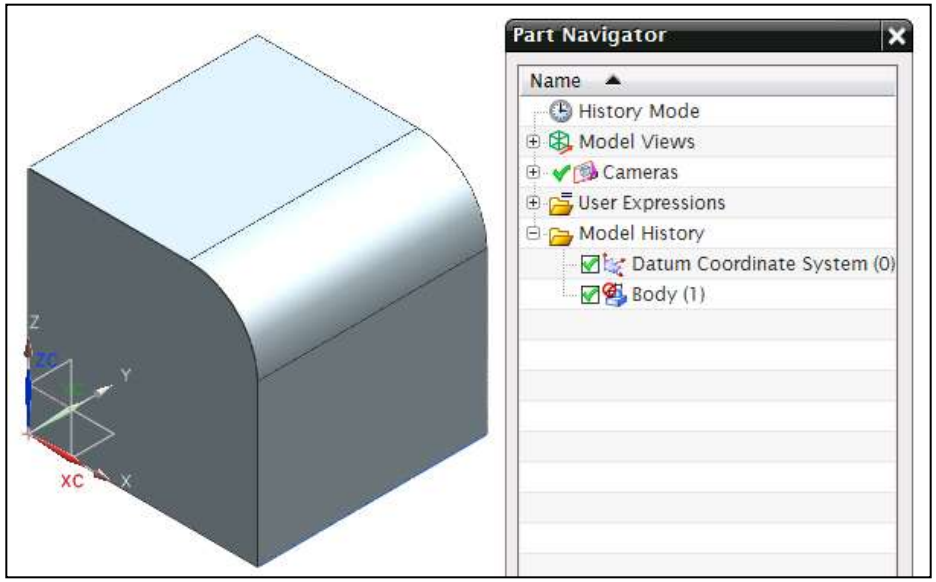
 (Trim Corner)

 (Trim Body)

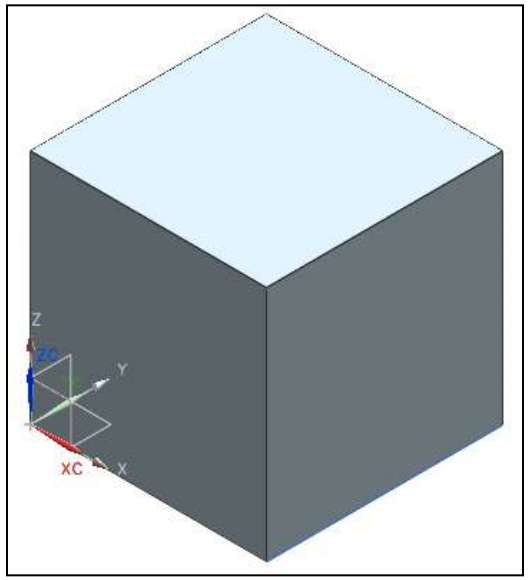
 (Trimmed Sheet)

6. 완성된 결과를 다시 진행해 볼 때는 (또 다른) 타 방법으로 해보려는 자세

예) Block의 Edge Blend를 Blend 없는 각진 Edge로 변경하는 또 다른 타 방법들

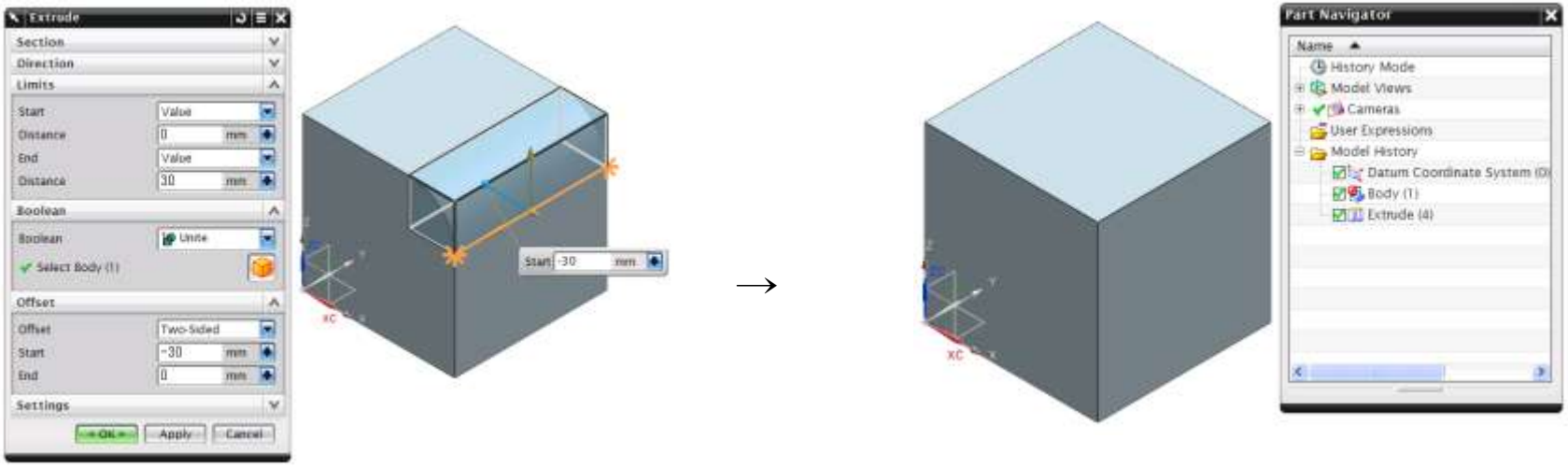


(Edge Blend 제거 전)

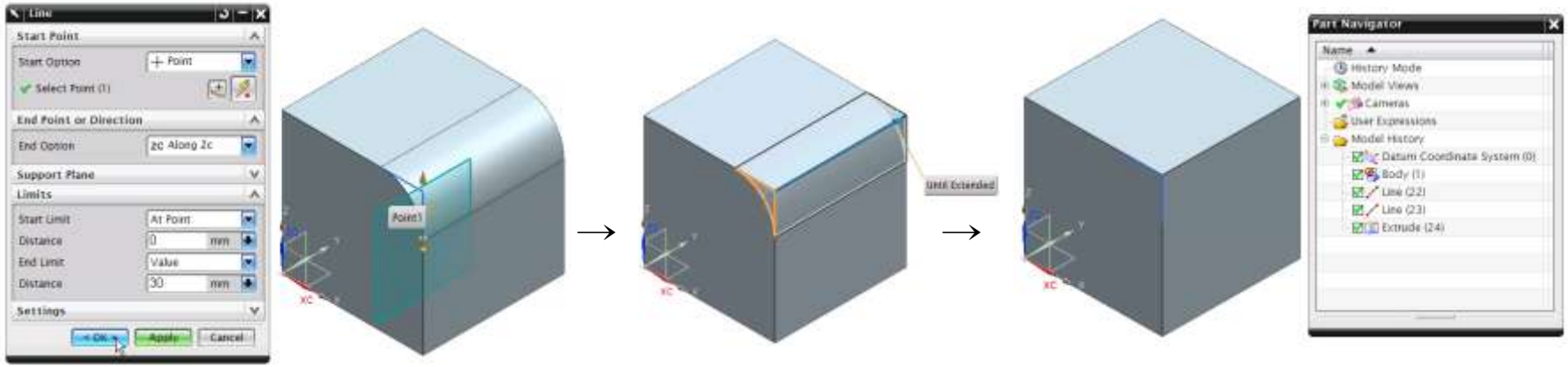


(Edge Blend 제거 후 결과)

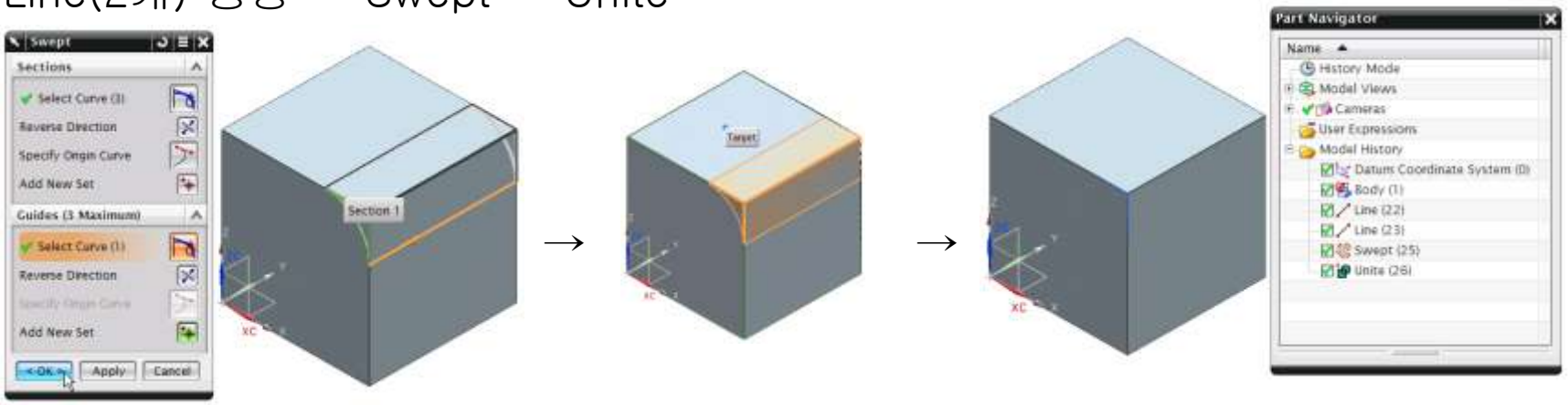
Block 자체 Edge(1개) 사용 → Extrude



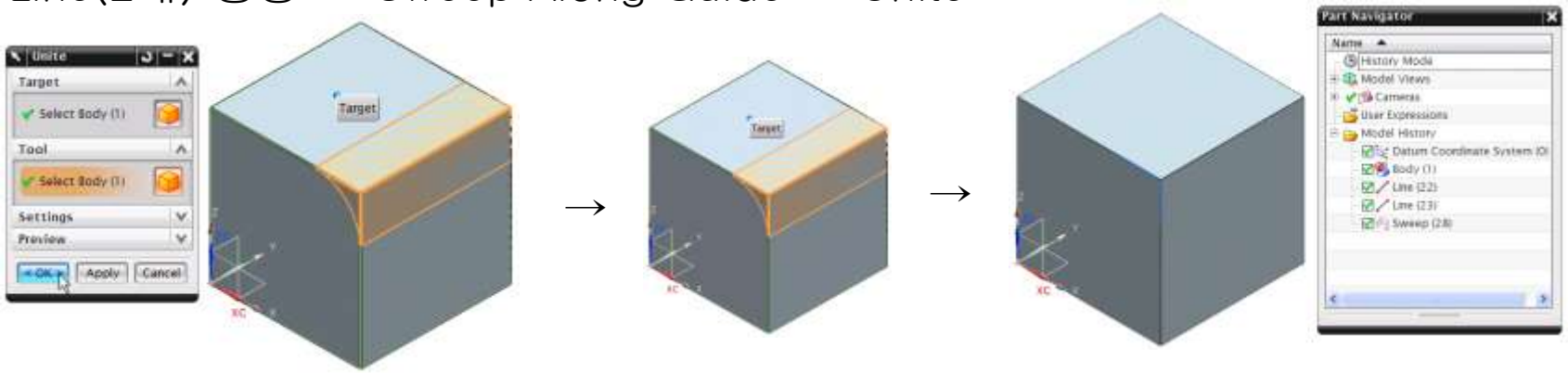
Line(2개) 생성 → Extrude



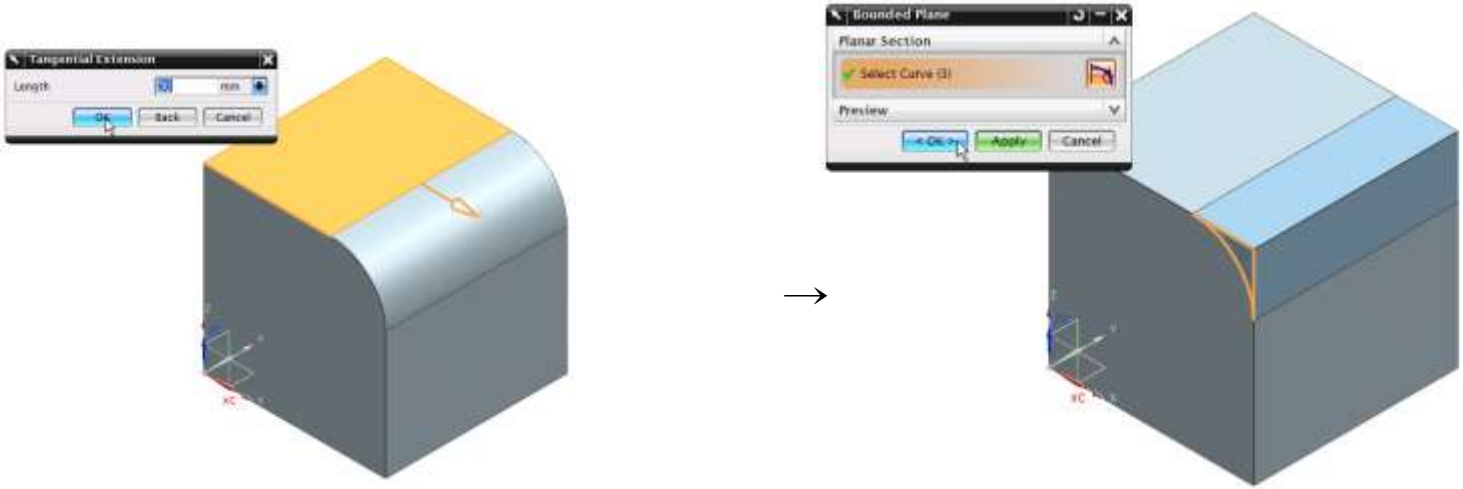
Line(2개) 생성 → Swept → Unite

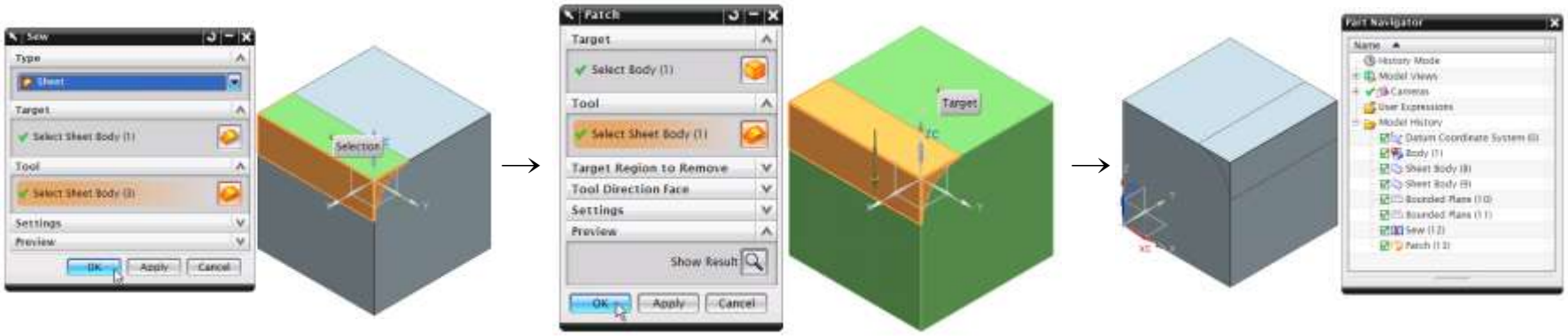


Line(2개) 생성 → Sweep Along Guide → Unite

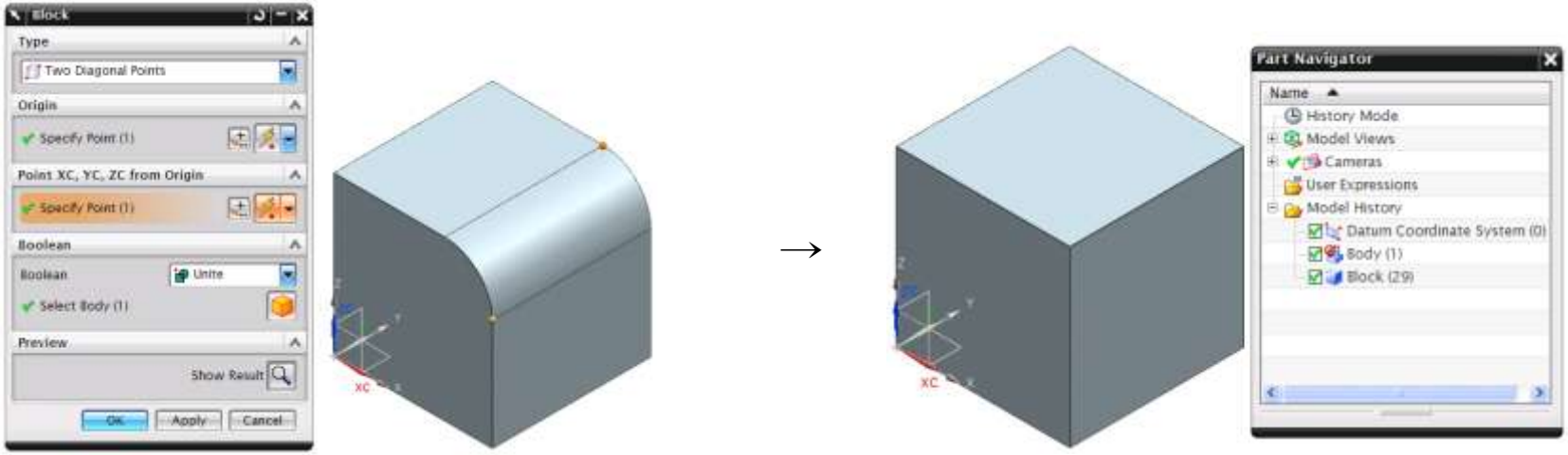


Extension(2개) 면 생성→ Bounded Plane(2개) → Sew → Patch

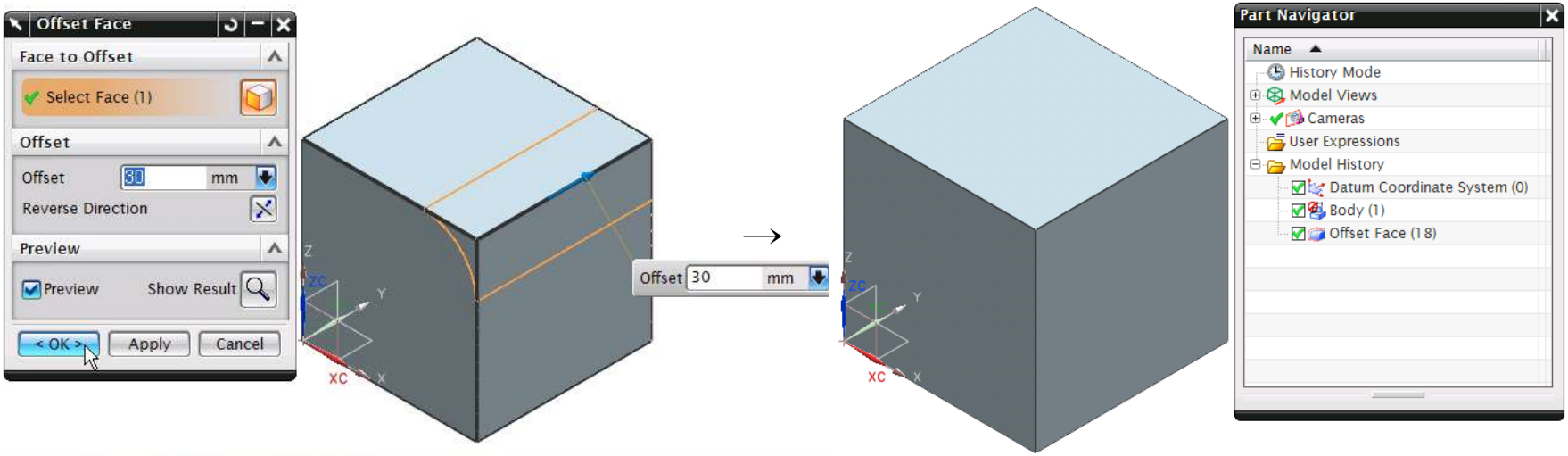




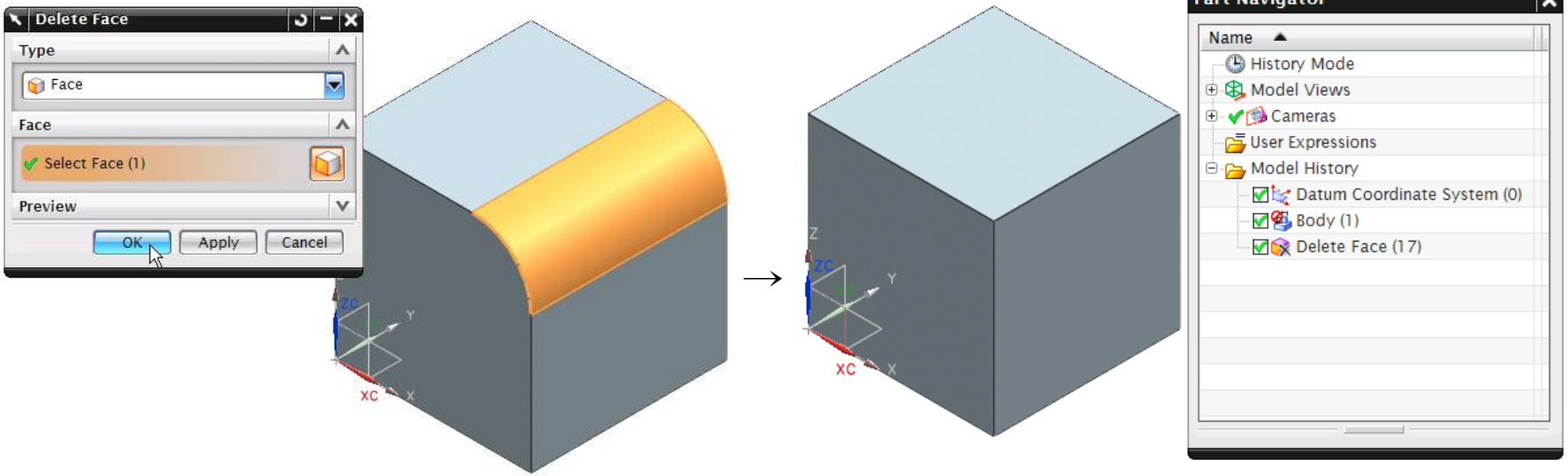
Block(Two Diagonal Points) → Unite



Offset Face

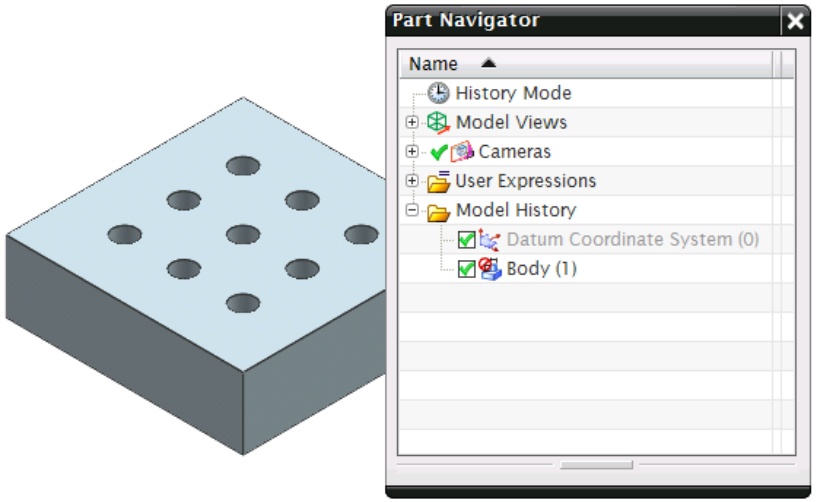


Delete Face(Synchronous Modeling)

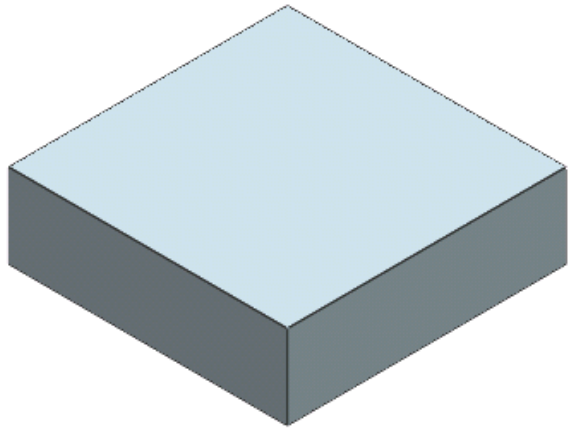


7. 배울 때는 힘들고, 다양하게! 실제 현업 작업은 쉽고, 간결하게!

예) Block의 Hole과 추출된 Sheet의 Hole을 제거할 수 있는 다양한 방법들



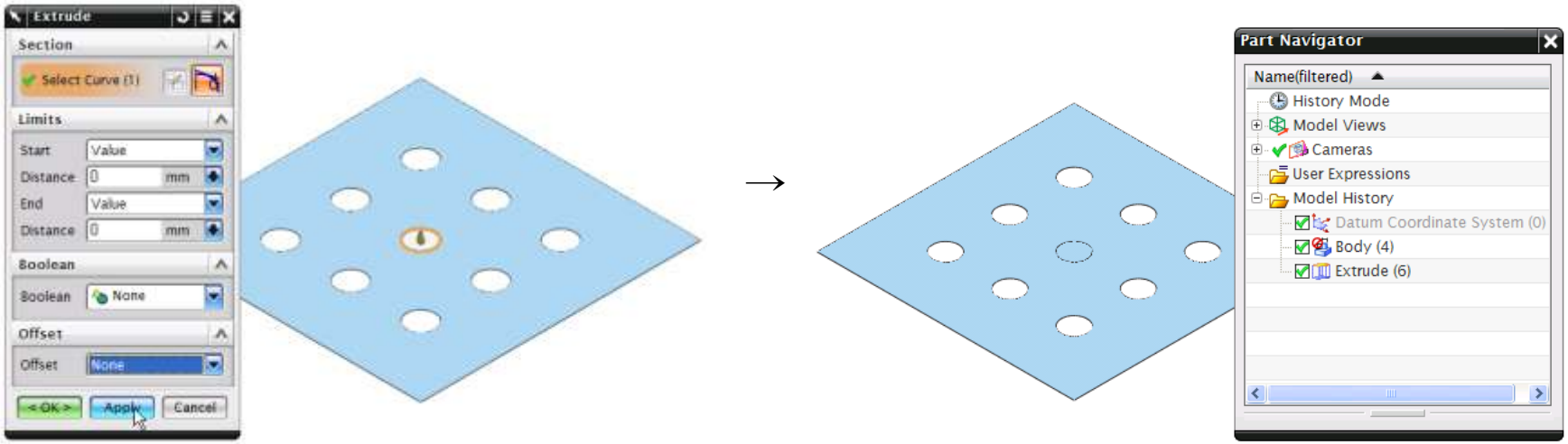
(Hole 제거 전)



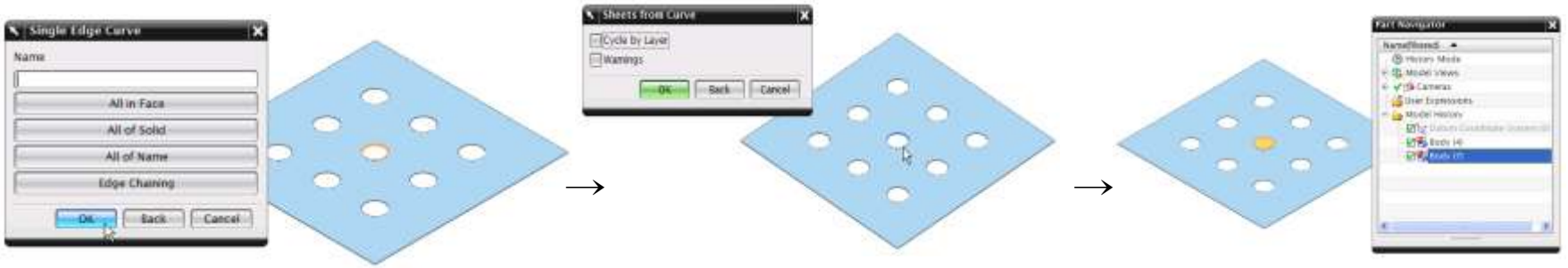
(Hole 제거 후 결과)

배울 때는 힘들고, 다양하게!

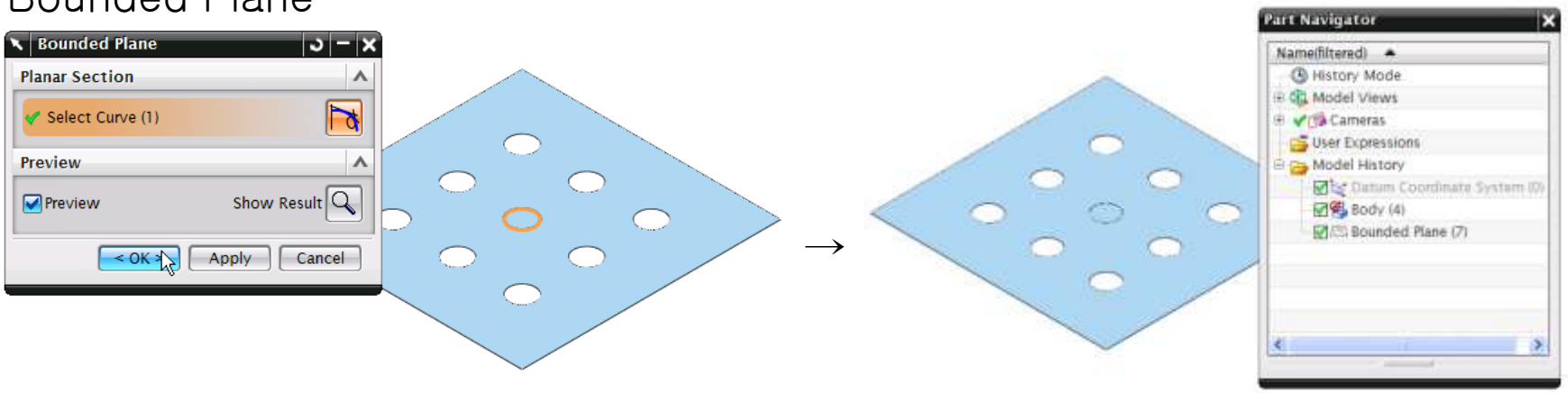
Extrude



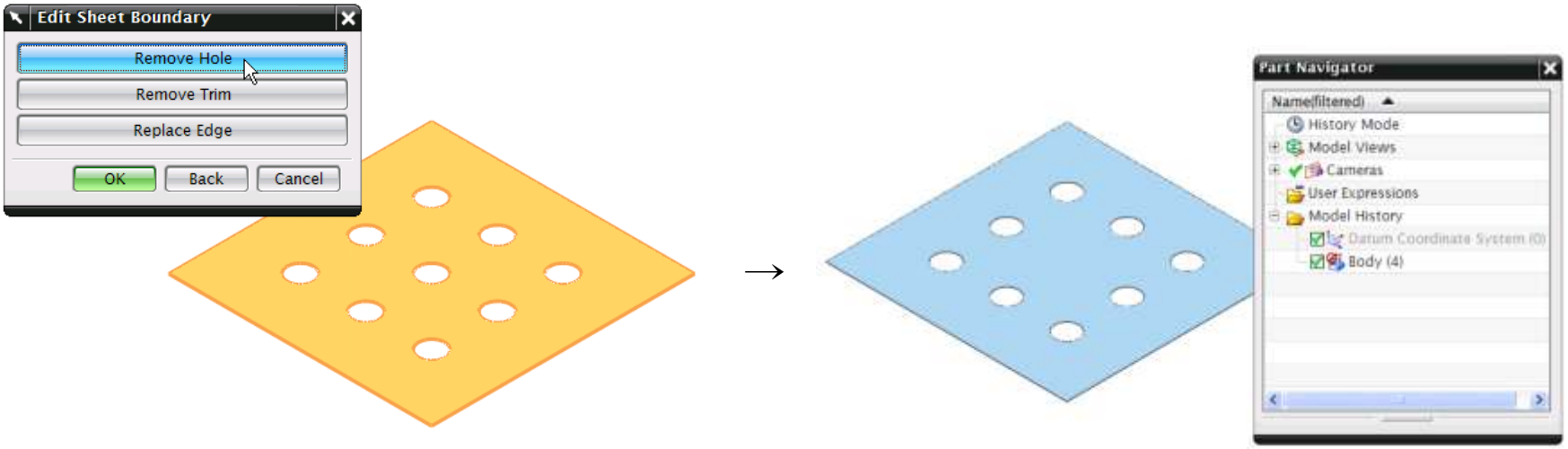
Extract Curve → Sheet From Curve



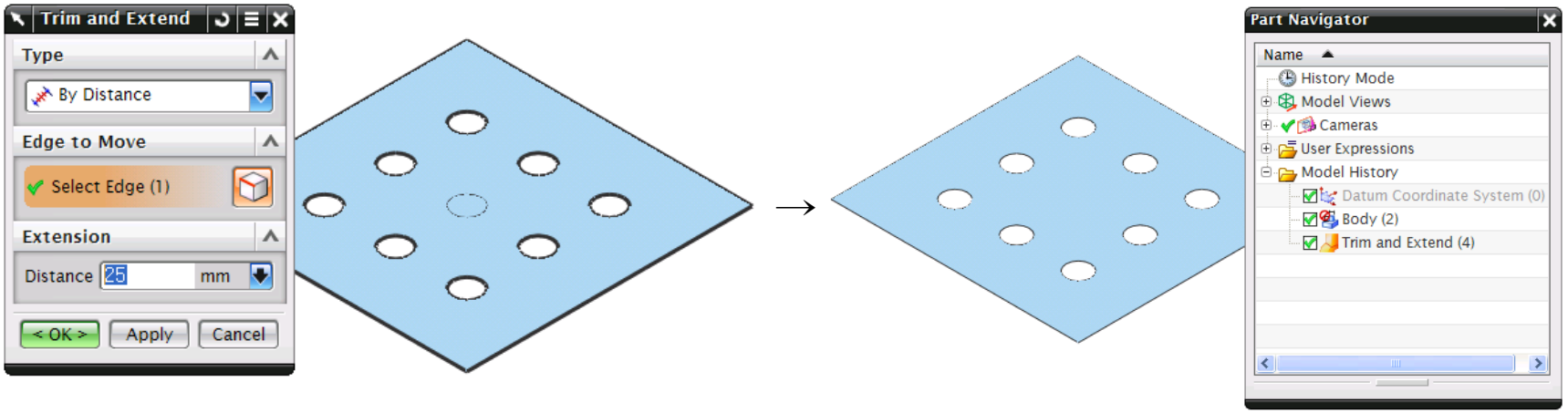
Bounded Plane



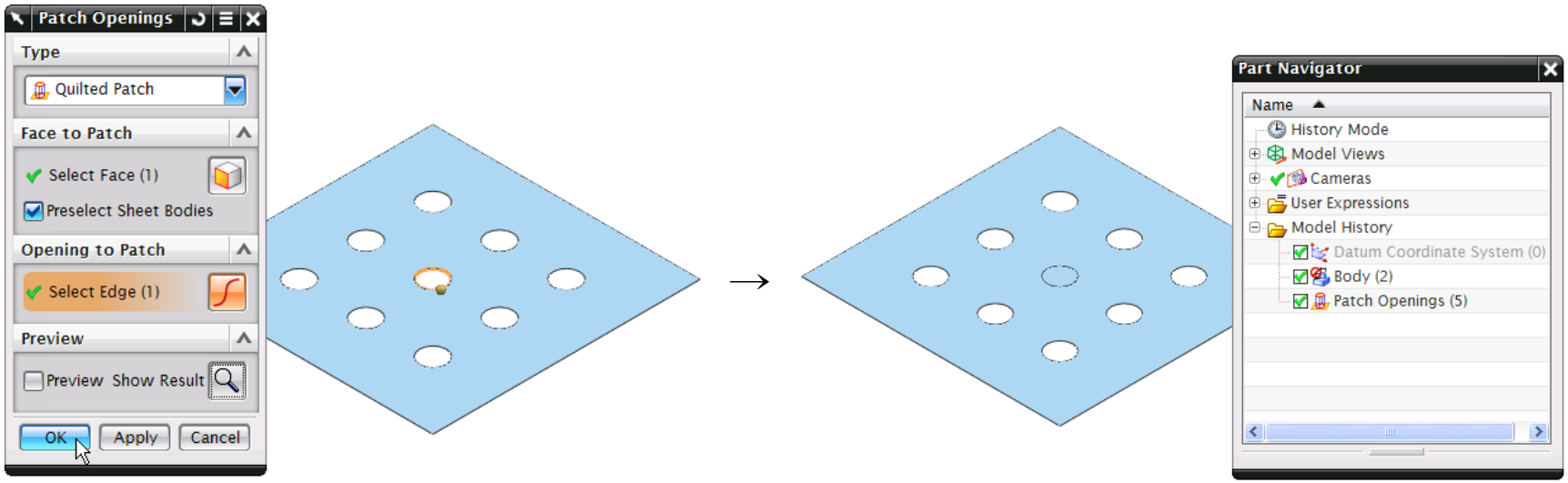
Edit Sheet Boundary(Remove Hole)



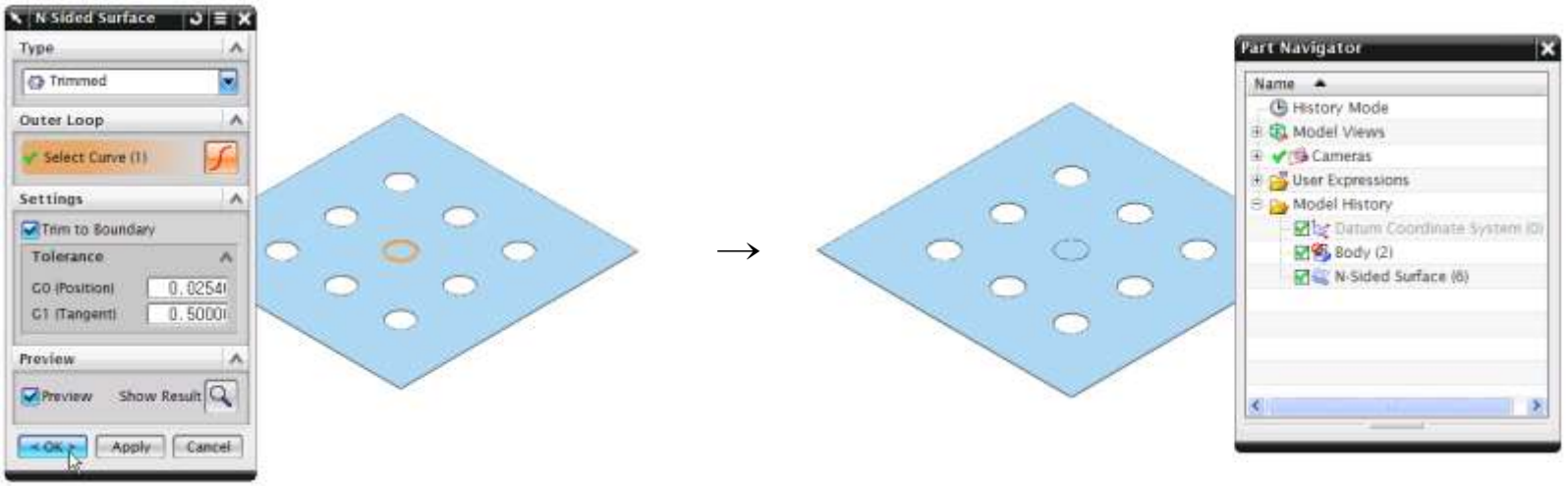
Trim and Extend



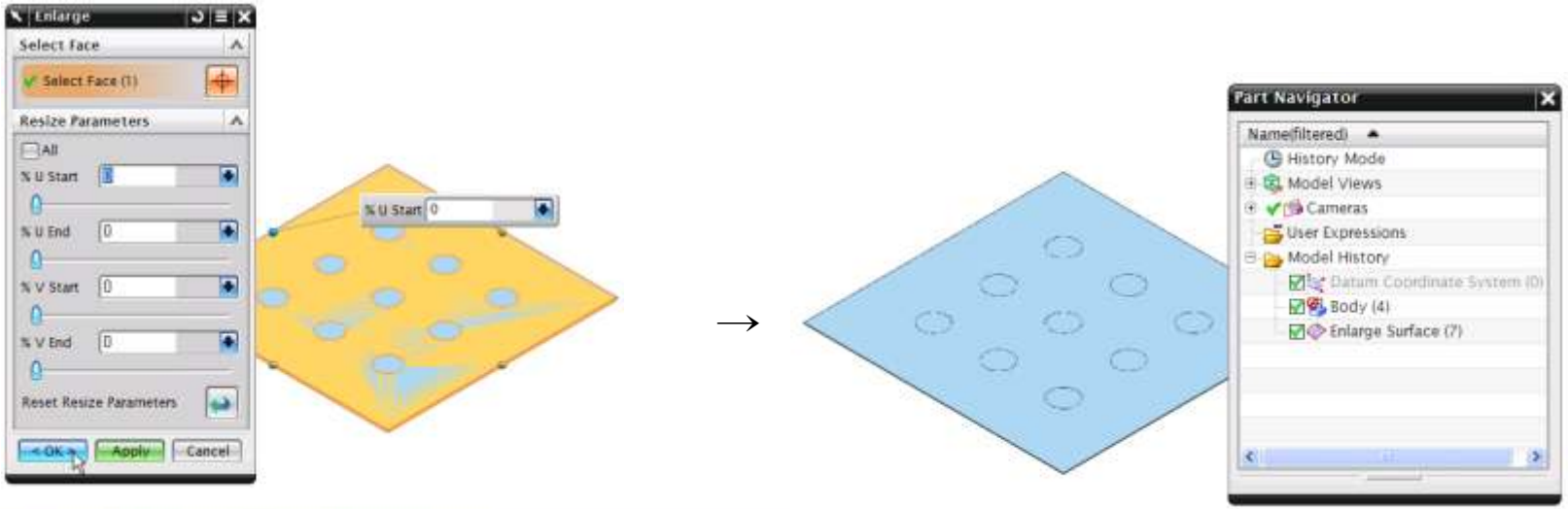
Patch Openings



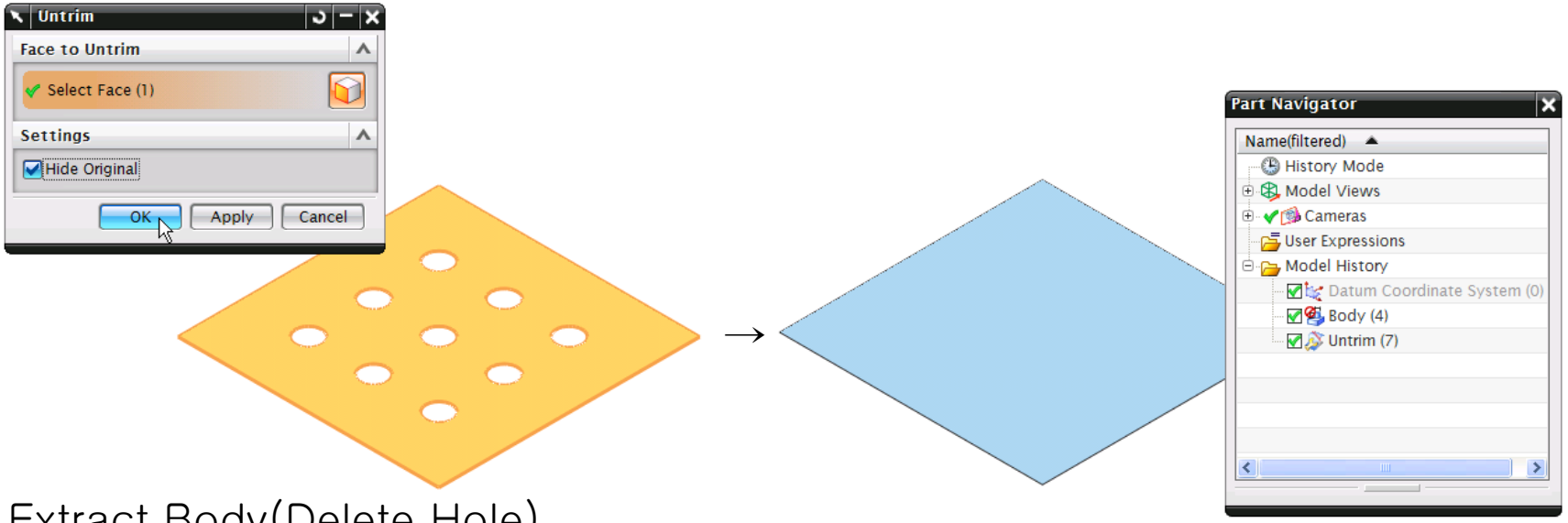
N-Sided Surface



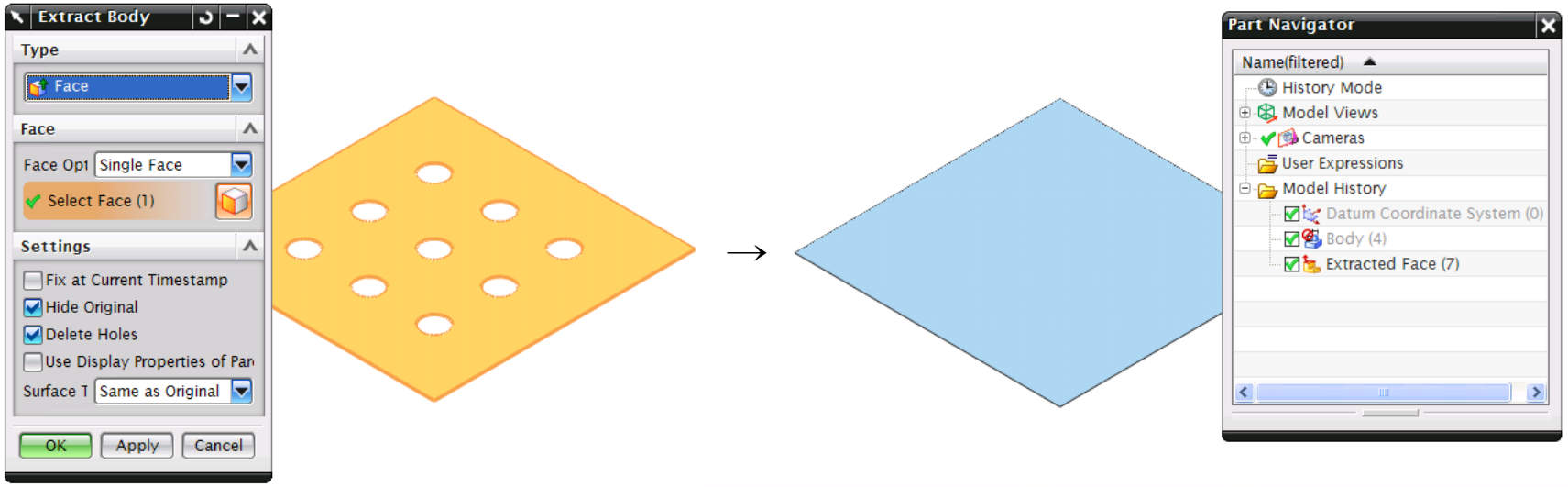
Enlarge



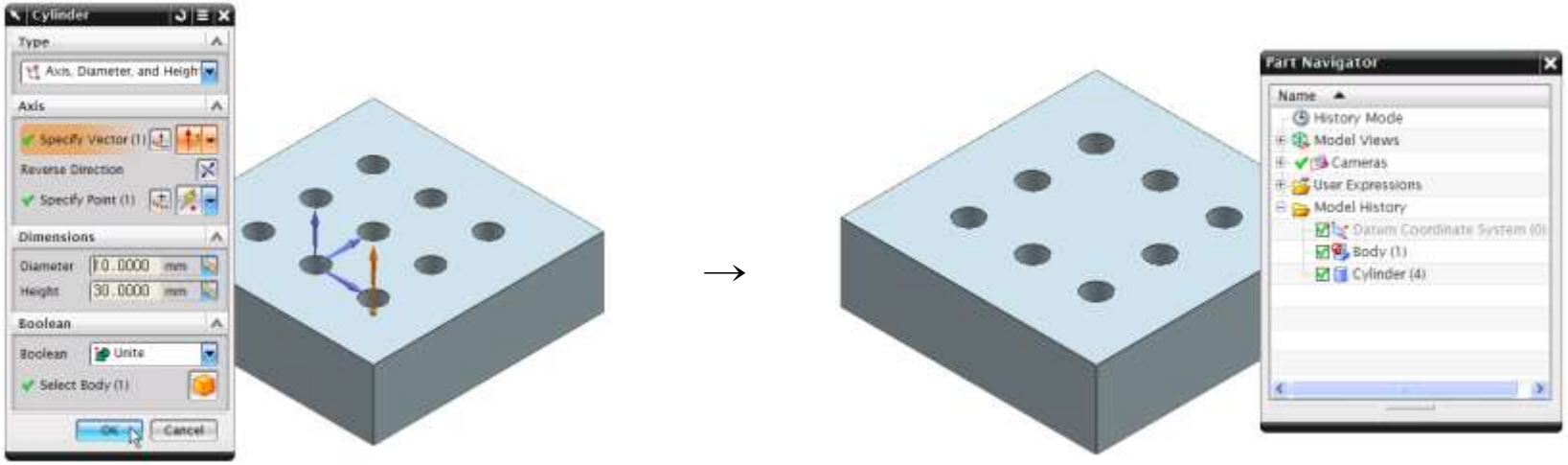
Untrim



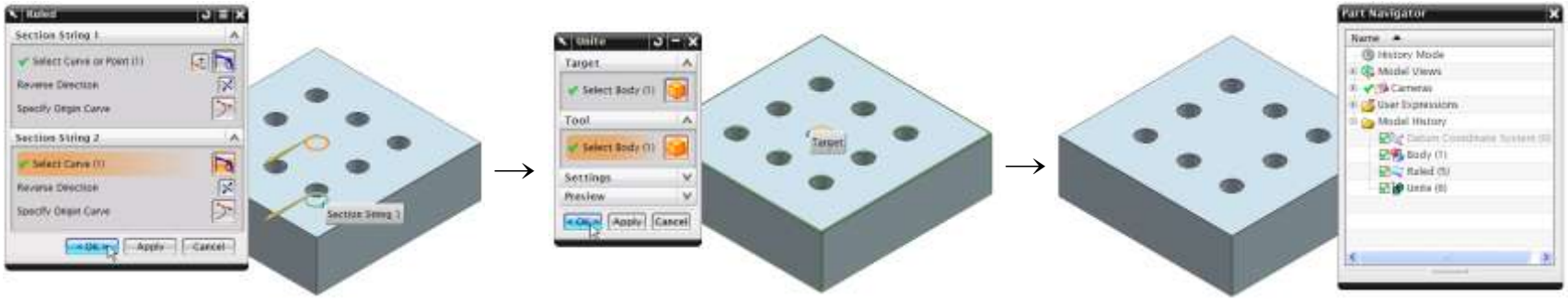
Extract Body(Delete Hole)



Cylinder/Unite

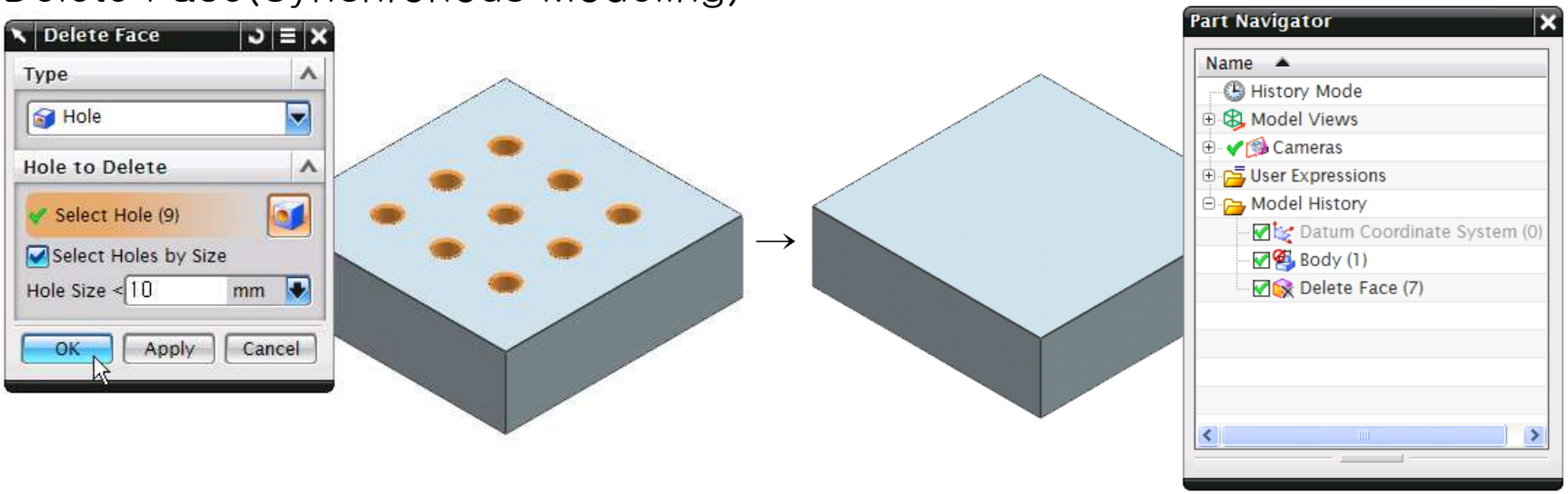


Ruled → Unite



실제 현업 작업은 쉽고, 간결하게!

Delete Face(Synchronous Modeling)



NX Modeling의 Step by Step Study 분류(대원캐드캠디자인학원: 60h/Step 예시)

Step1_Study(NX 일반모델링)

- NX Windows/Roles/Modeling Preferences/Ctrl-Alt-Shift-Enter Key
- File과 관련된 제반 사항/Mouse Button/Navigator
- Quick Pick/Radial Pop-up/Quick View Pop-up/Selection (Mini)Bar
- Function/Shortcut key(Ctrl+D, Ctrl+J, Ctrl+I, Ctrl+W, Ctrl+B, Ctrl+T)
- Datum Plane/Datum Axis/Datum CSYS/Point-Edit Point
- ABS/WCS(Dynamics, Orient, Rotate, etc)
- History Mode/History-Free Mode Comparison
- Sketch(Make Corner, etc)/Edit Sketch(Attach Dimension, etc)
- Transform(Ctrl+T)/Move Object(Ctrl+Shift+M)
- Spline/Studio Spline/Basic Curve Fillet
- Format-Layer(Category, Setting, Copy, Move), Visible in View
- Primitive Feature(Block, Cylinder, Cone, Sphere)/Tool-Expression
- Boolean Operation(Unite, Subtract, Intersect)
- Pre-NX5 Hole/Hole/Boss/Pocket/Pad/Slot/Groove/Dart
- Edit Positioning/Edit Feature Parameters(Reattach)/Edit with Rollback
- Extrude/Edge Blend/Chamfer/Revolve/Draft/Draft Body/Shell
- Trim Body/Split Body/Patch
- Offset Face/Scale Body
- Instance Feature(Rectangular Array, Circular Array, Pattern Face)
- Mirror Body/Mirror Feature/Mirror Curve/Wrap Geometry
- Delete Face(Hole to Delete) VS Extract Geometry(Delete Holes)
- Replace Feature/Reorder Feature/Move Feature
- Remove Parameters/Suppress/Unsuppress Feature/Suppress by Expression
- Copy Feature/Paste Feature(Create New/Link to Original/Reuse Oringinal)
- Copy Display/Update Model/Edit Solid Density/(Automatic)Feature Playback
- Ruled/Through Curves/Through Curve Mesh/Swept
- Variational Sweep/Sweep Along Guide/Tube
- Thicken Sheet/Sheets to Solid Assistant
- Analysis(Distance/Angle/Arc Length/Measure Bodies/Shape Face Reflection)
- General Assembly(Top-Down/Bottom-up Method)
- Assembly-Move Component/Check Clearance
- Assembly Cut/Simple Interference
- General Drafting
- Import, Export(DXF/DWG, IGES, CGM, JPEG, etc)

Step2_Study(NX 곡면모델링)

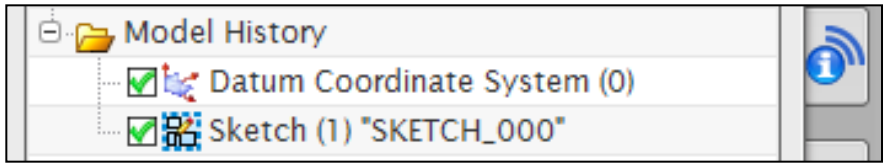
- Curve Chamfer/Rectangle/Polygon/Ellipse
- General Conic/Helix/Law Curve
- Line
- Arc/Circle
- Lines and Arcs
- Edit Spline/Fit Spline/Smooth Spline/Shape by Template
- Trim Curve/Trim Corner
- Divide Curve/Edit Fillet
- Arc Length/Edit Curve Length
- Thread/Make Current Feature
- Stretch Curve/Offset Curve/Curve on Surface/Offset in Face
- Bridge Curve/Circular Blend Curve cf. Two Curve Fillet, General Conic(2 Points, 2 Slope, Rho)
- Extract Curve
  (Edge Curves, Isoparametric Curves, Silhouette Curves
  All in Work View, Isocline Curves, Shadow Outline)
- Composite Curve/Join Curves/Simplify Curve
- Project Curve/Combined Projection
- Wrap/Unwrap Curve
- Intersection Point/Intersection Curve/Section Curve
- Extract Geometry
- Divide Face/Join Face
- Four Point Surface/Emboss Sheet/Bridge
- Swoop/Reverse Normal/Midsurface/Transition
- Ruled/Through Curves/Through Curve Mesh/Swept
- Studio Surface/Styled Sweep
- Trimmed Sheet/Untrim
- Trim and Extend
- Sew/Unsew
- Face Blend/Soft Blend/Styled Blend/Aesthetic Face Blend
- Patch Openings
- Edit Sheet Boundary/N-Side Surface
- Bounded Plane/Sheet from Curves
- Enlarge/Extension/Law Extension
- Silhouette Flange
- Offset Surface/Variable Offset Surface/Rough Offset
- Analysis(Shape/Minimum Radius/Deviation/Measure Face)

Step3_Study(NX 고급)

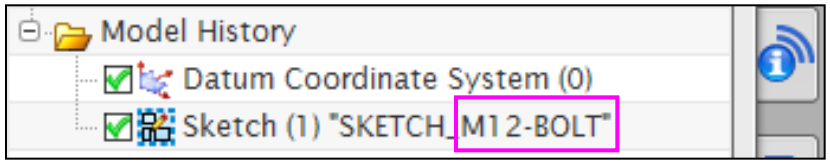
- Basic Curves(Line/Arc/Circle/Trim)
- Edit Curve Parameters/Text/Point Set
- Through Points/From Poles/From Point Cloud
- Deform Sheet/Transform Sheet/X-Form/I-Form/Quilt
- Match Edge/Refit Face/Pole Smoothing/Edge Symmetry
- Change Edge/Change Degree/Change Stiffness
- Snip Surface/Isoparametric Trim/Divide
- Emboss/Offset Emboss/Move Pole/Defining Point
- Global Shaping by Function/Global Shaping by Surface
- Styled Corner/Spherical Corner/Ribbon Builder
- Section Surface(1~20 Types)
- User Defined Feature(UDF)
- Instance Geometry
- Reuse Library
- External Multi-CAD(CATIA, Pro-E) Model Component Converting
- Synchronous Modeling(History Mode/History Free Mode)
  Move Face/Pull Face/Offset Region/Replace Face/Resize Blend/Resize Face
  Delete Face/Group Face/Cross Section Edit
  Chamfer_Resize Chamfer/Label Chamfer
  Reuse_Copy Face/Cut Face/Paste Face/Mirror Face/Pattern Face
  Constrain_Make Coplanar/Make Coaxial/Make Tangent/Make Fixed
  Make Symmetric/Parallel/Perpendicular/Offset/Show Related Face
  Dimension_Linear Dimension/Angular Dimension/Radial Dimension
  Shell_Shell Body/Shell Face/Change Shell Thickness
  Optimize_Optimize Face/Replace Blend
- Promote Body/Wave Geometry Linker/Replace with Independent Sketch
- Assembly Sequences(Extraction Path/Motion Envelope)
- Assembly Explode/View Save-Drafting Parts List-Mass/Volume Check
- Utilities-Part Cleanup/Character Fonts/Create Custom Symbol
- Advanced Drafting(Part View, Drawing View, View Dependent Edit
  Parts List-Auto Balloon, Ordinate-Edit Ordinate, Hole Table, Broken View)
- PMI(Product Manufacturing Information) - Drafting Inherit PMI
- Analysis(Units Manager/Model Compare/MPV/Space Finder)
- Shape Studio(Rendering)
- Motion Simulation(Synchronized Mechanism Dynamic Analysis)

8. NX 수정작업/협업작업 시 함께 하는 “사람”을 배려하는 마음으로 !

상대방을 배려하는 마음으로: Sketch 이름에 “M12-Bolt”를 추가한 예

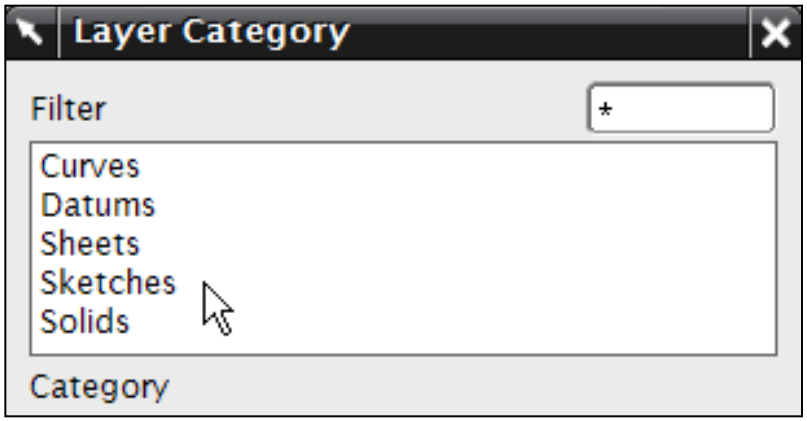


(기본 설정)

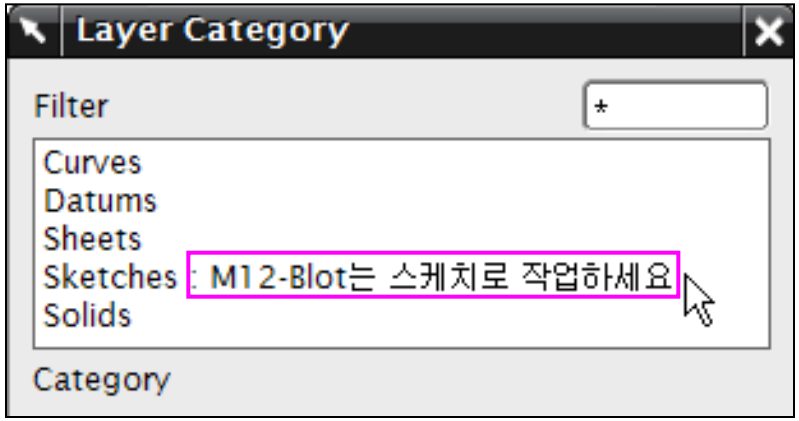


(추가 설명)

상대방을 배려하는 마음으로: Layer 범주에 “M12-Bolt”를 추가 설명한 예



(기본 설정)



(추가 설명)

상대방을 배려하는 마음으로: 약어보다 사용한 Tool Type과 직경을 설명한 예

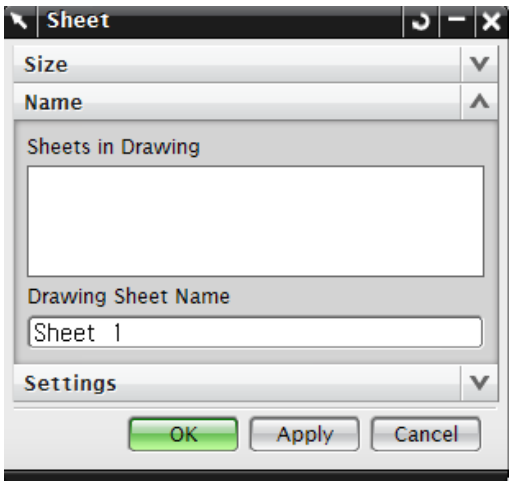
Name	Tool	Tool Description	Tool Number	Path
NC_PROGRAM				
Unused Items				
PROGRAM				
MANUFACTURING.NC				
CAVITY_MILL	T01	FEM12	1	✔
FIXED_CONTOUR	T02	BEM6	2	✔
FLOWCUT_REF_TOOL	T03	BEM2	3	✔

(약어 설명)

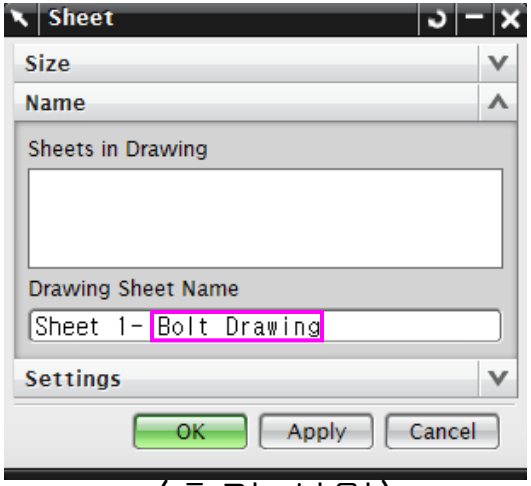
Name	Tool	Tool Description	Tool Number	Path
NC_PROGRAM				
Unused Items				
PROGRAM				
MANUFACTURING.NC				
CAVITY_MILL	T01_FEM12	T01_Flat-End-Mill_12mm	1	✔
FIXED_CONTOUR	T02_BEM6	T02_Ball_End-Mill_6mm	2	✔
FLOWCUT_REF_TOOL	T03_BEM2	T03_Ball-End-Mill_2mm	3	✔

(상세 설명)

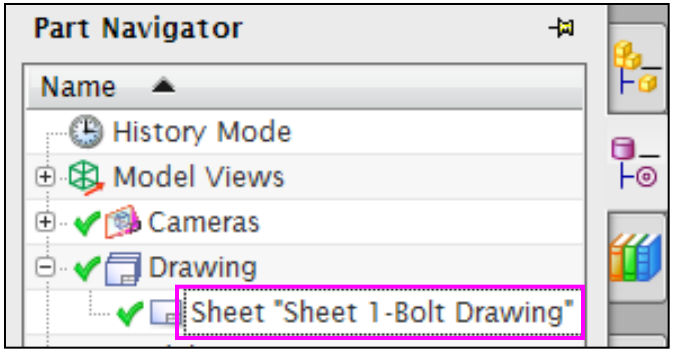
상대방을 배려하는 마음으로: 사용 Sheet1이 “Bolt-Drawing”이라는 추가 설명 예



(기본 설정)



(추가 설명)



(Navigator에 반영된 설명)

9. NX Veteran/Master의 필수어(Fast_Exactly_No History/3S/3R)

Fast(작업은 빠르게!)
Exactly (작업은 정확하게!)
No History(작업한 History는 없거나 적게!)

3S_Campaign
Simplification(누구나 도면/Navigator 정보를 보면 쉽게 알 수 있도록 단순화)
Standardization(어떤 국가에서도 상호 소통될 수 있는 제품/작업의 표준화)
Specialization(아무도, 어떤 국가에서도 따라오기 어려운 자신만의 전문화)

3R_Campaign
Reduction(작업 시간/재료 등등의 절감)
Reuse(동일한 부품의 반복 작업시간을 줄이기 위한 표준 Componet의 재사용)
Recycle(Life Cycle이 종료된 Componet의 재활용/재순환)

NX 작업시 사용되는 마우스 같은 주변기기들을 내손처럼 사용!

10. (High End 급이라 불릴 만한) Good Software(S/W)는 어떤 것일까?

★ 구입가격에 비례해 S/W와 쉽게 친숙해질 수 있고, 안정성과, 사용하기 쉬워야 한다.

★ S/W가 제안해준 모의 Simulation의 결과와 실제 작업간의 최소 오차가 보증되어야 한다.

★ 사용자의 요구가 신 버전에 반영되며, 유지보수 잘되고, 비교 우위의 장점들이 다양하다.

★ PC의 새로운 환경체제의 변화에 따른 신속한 적응성과 호환성이 담보되어야 한다.

★ Dialog Box가 이해하기 쉽도록 그림으로 된 설명(Legend) 체계와 Help가 많아야 한다.

★ 버그(Bug)가 최소화되어 연산도중 PC의 다운 횟수가 적고 오류제거가 쉬워야 한다.

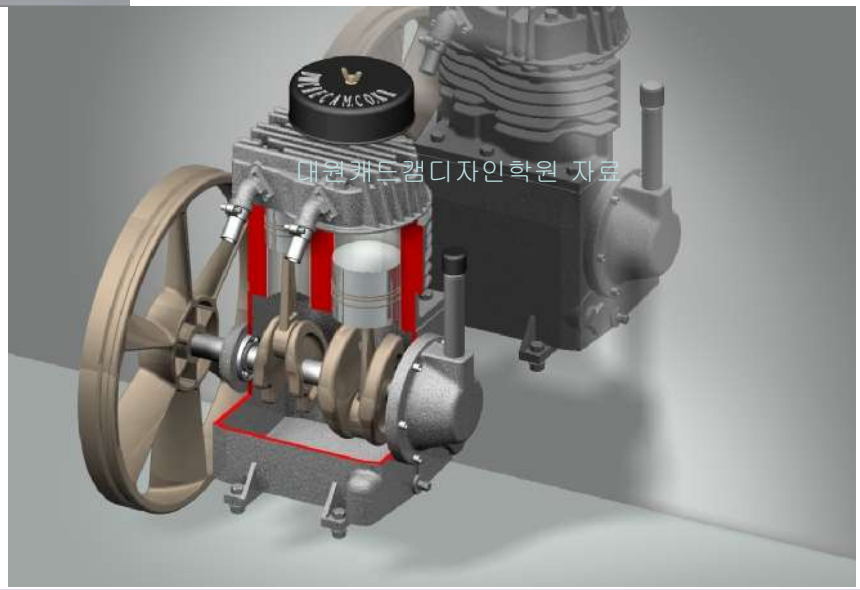
★ 설계 오류의 발생 원인이 즉시 진단되고 해결 정보를 문자나 음성으로 제공해야 한다.

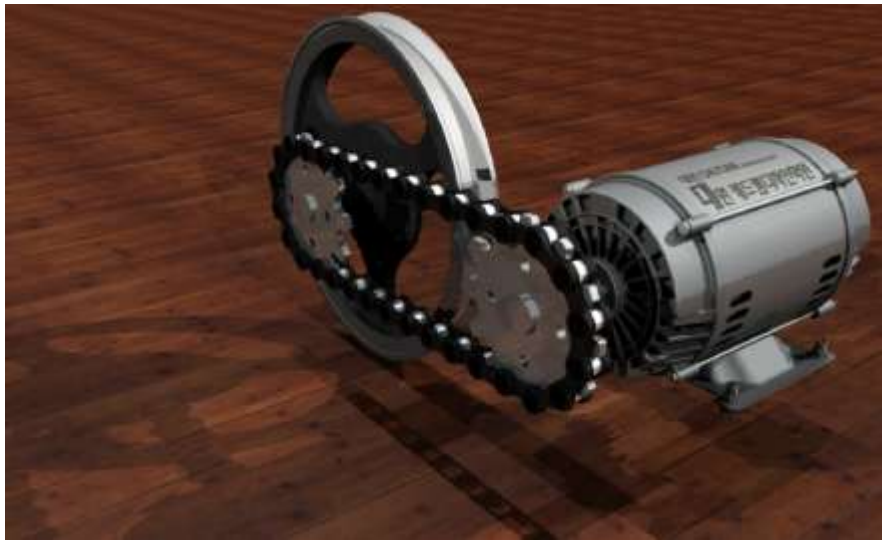
★ 표준 부품들을 재사용 할 수 있는 2D/3D Template Library가 많이 제공되어야 한다.

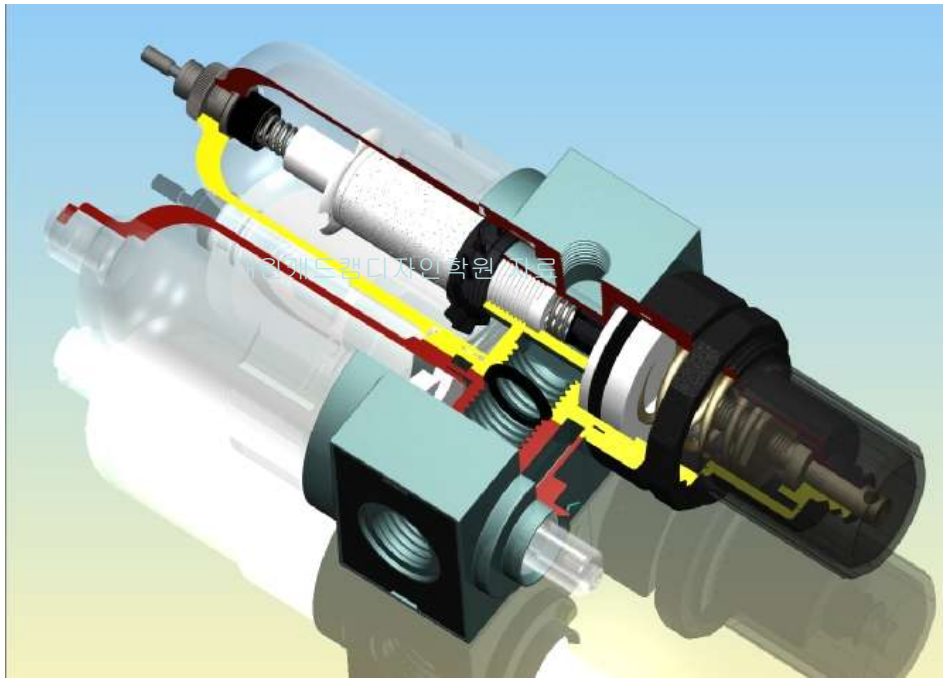
★ 음성인식 기술과 S/W를 사용하는 사용자의 휴대폰으로도 소통되는 기술이 제공된다.

★ 이종 소프트웨어 간 파일의 완벽한 변환/복사 및 World User가 많을수록 좋은 S/W 이다.

대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료



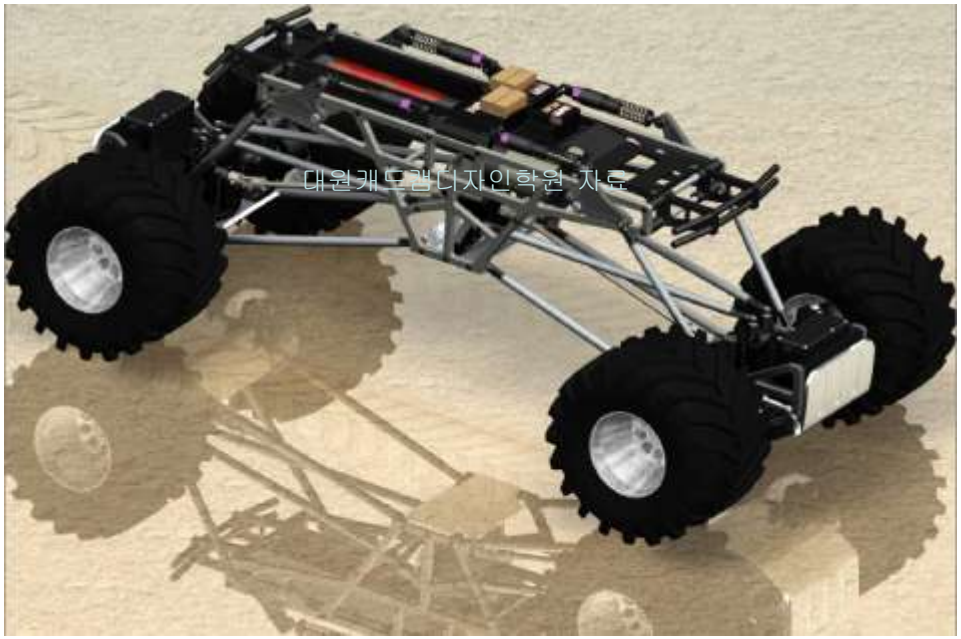



대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료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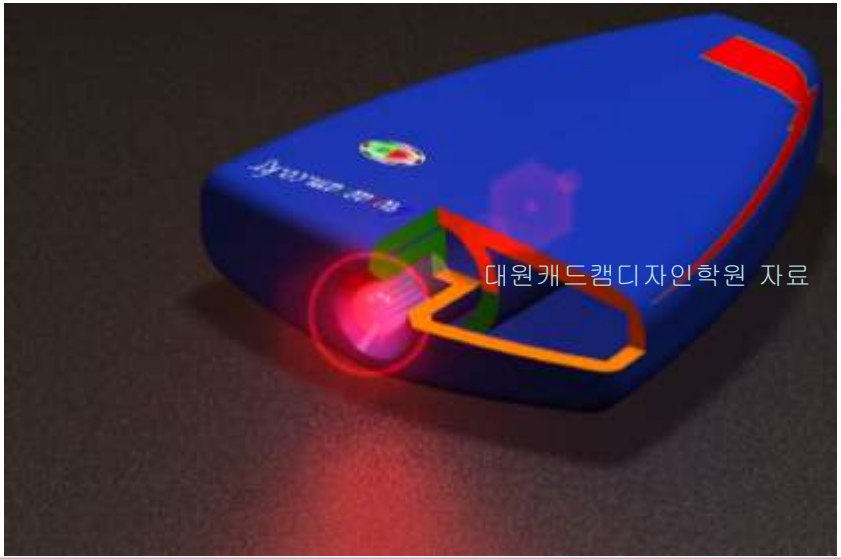
대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료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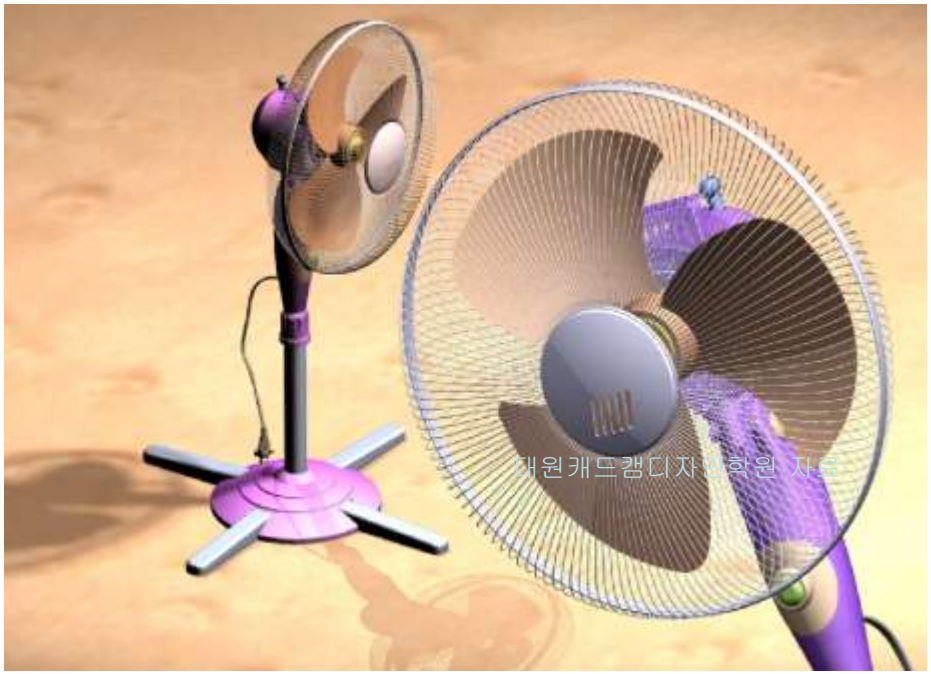
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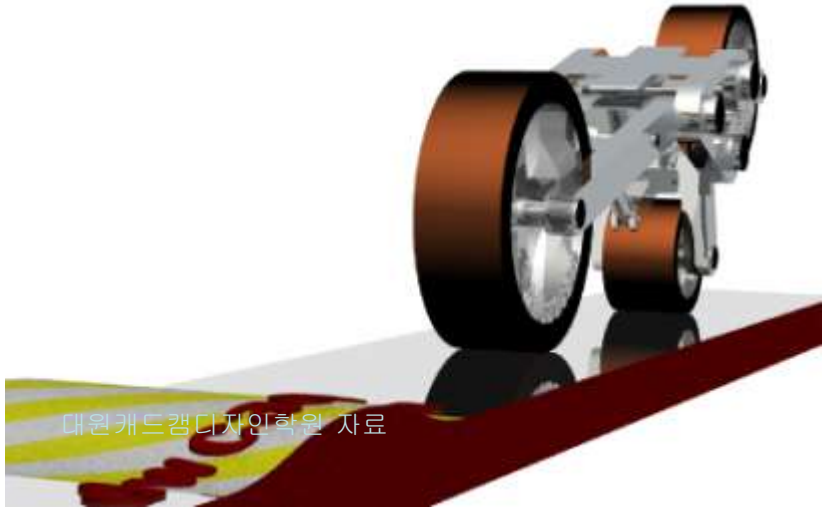
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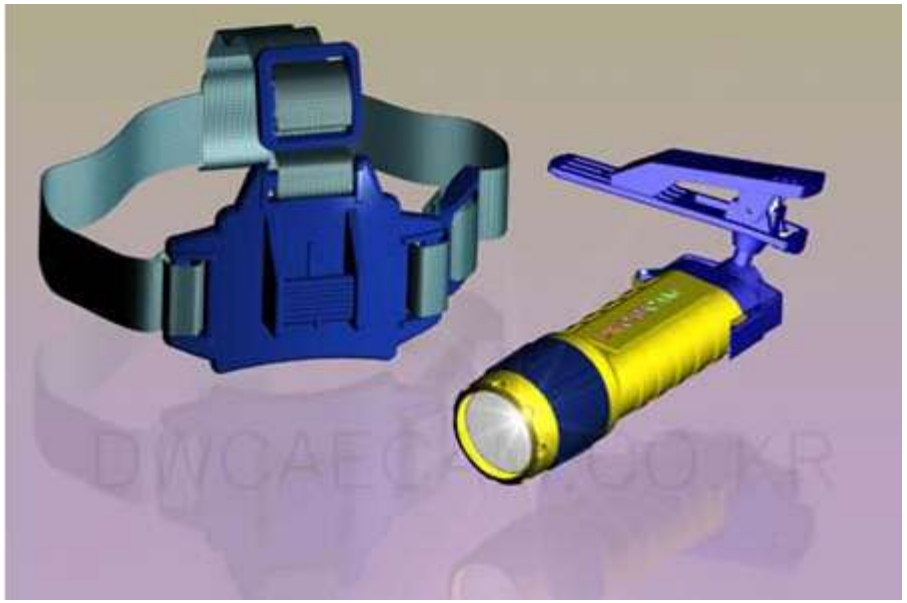

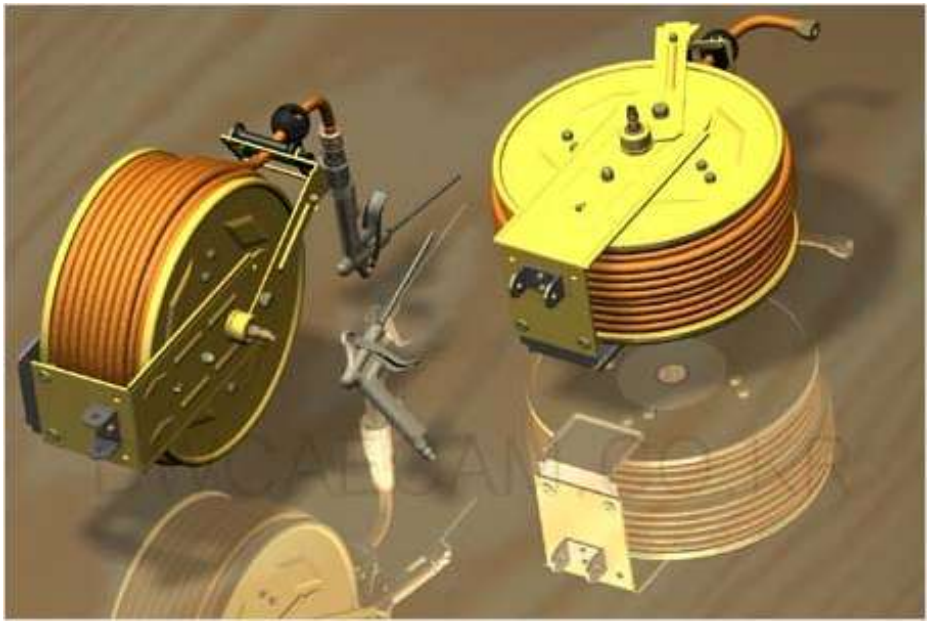


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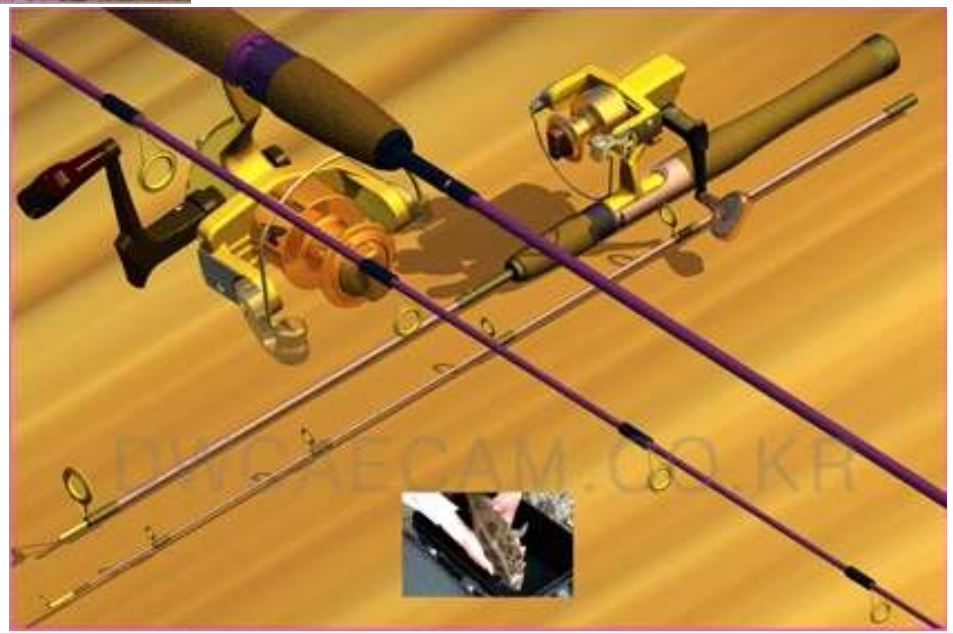
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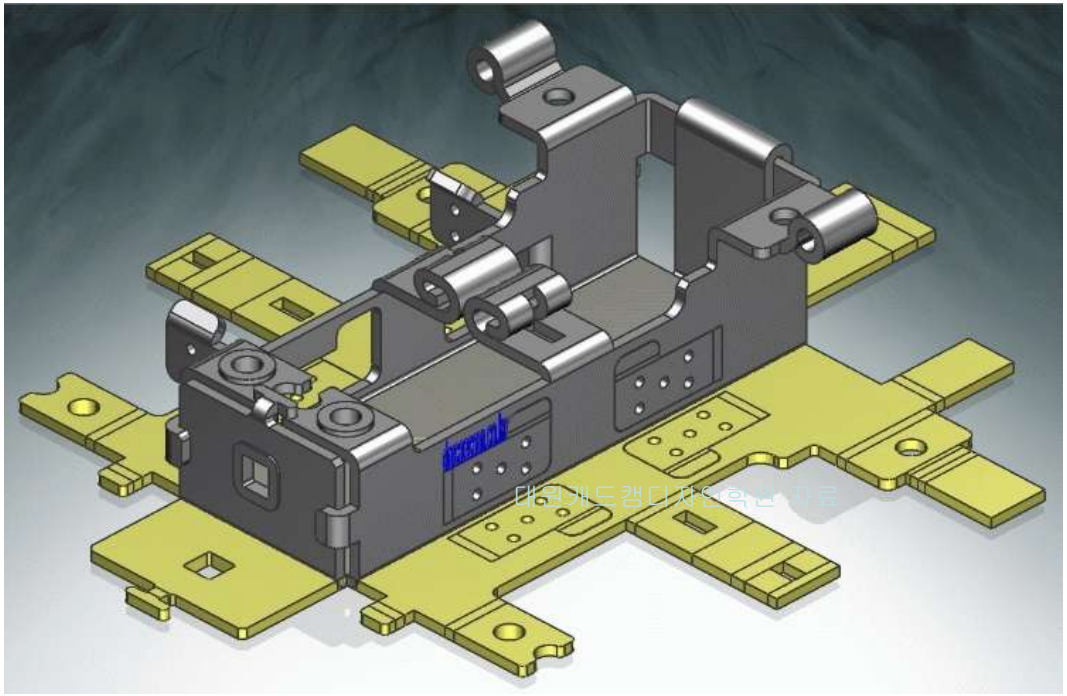
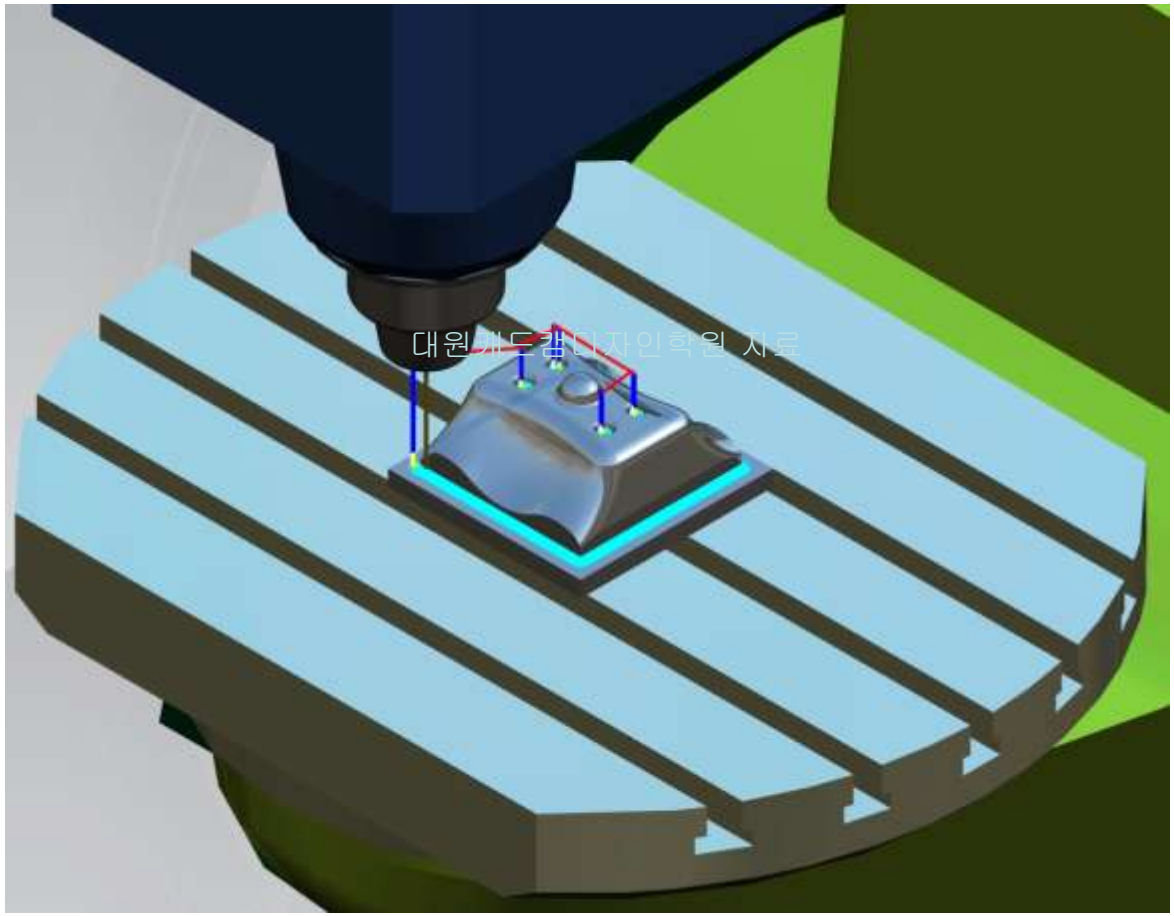
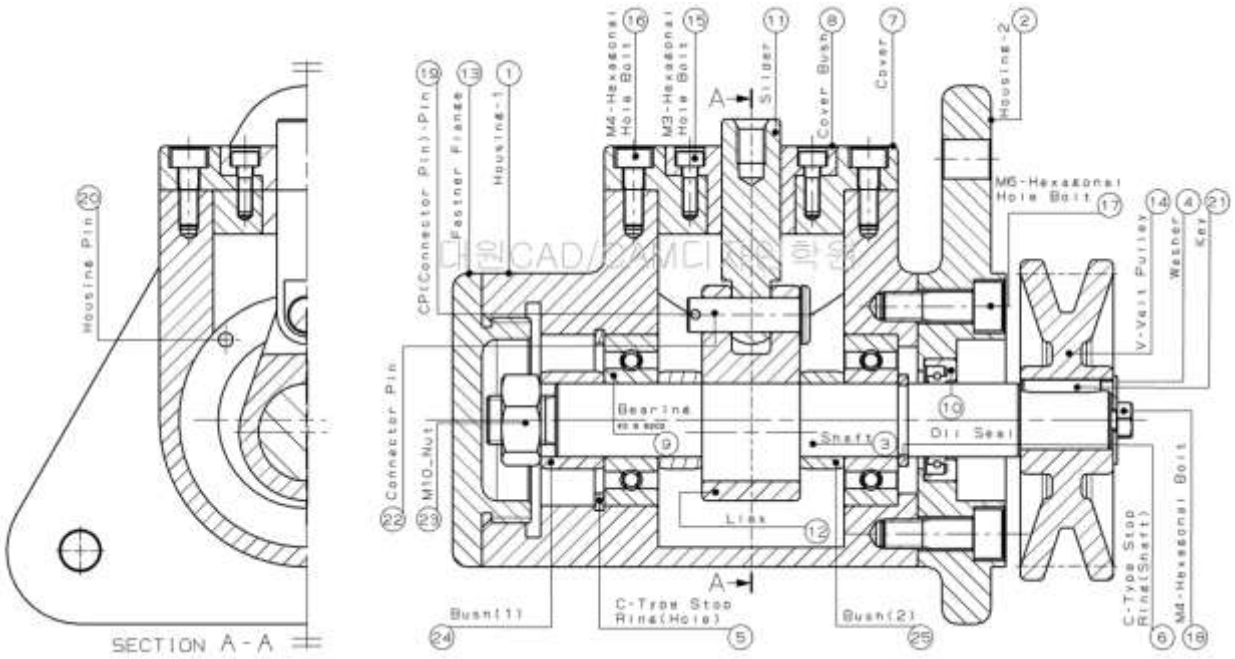
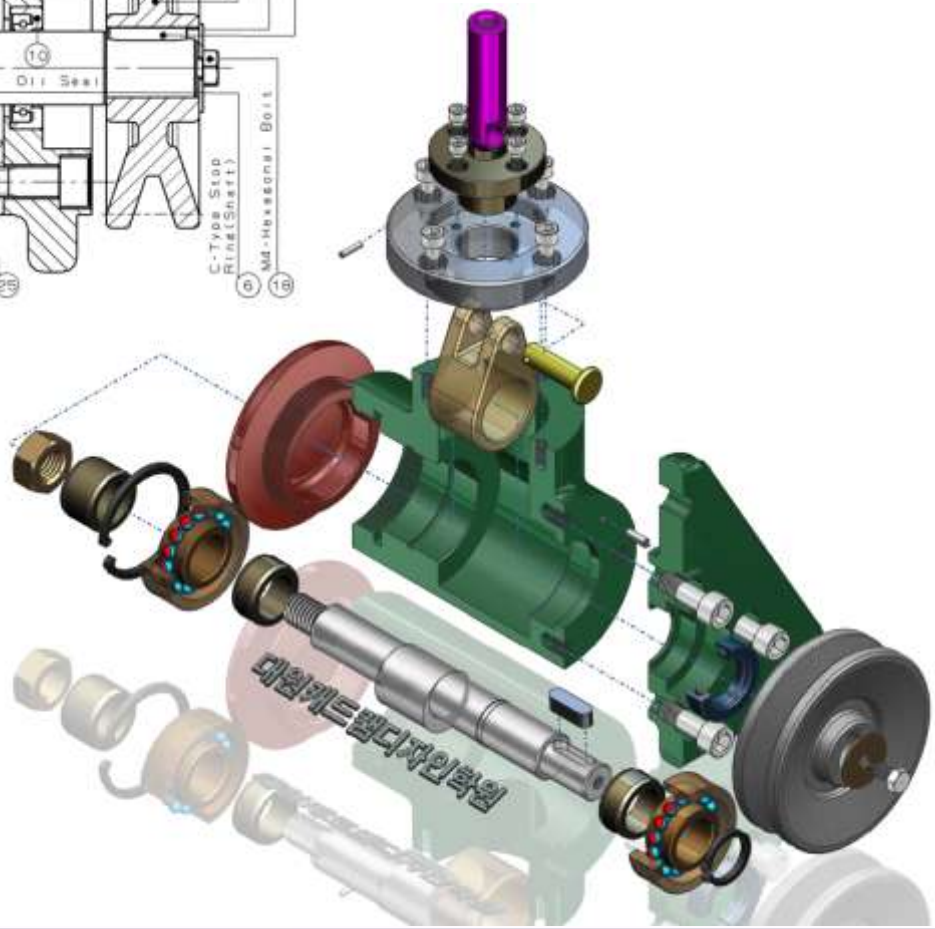
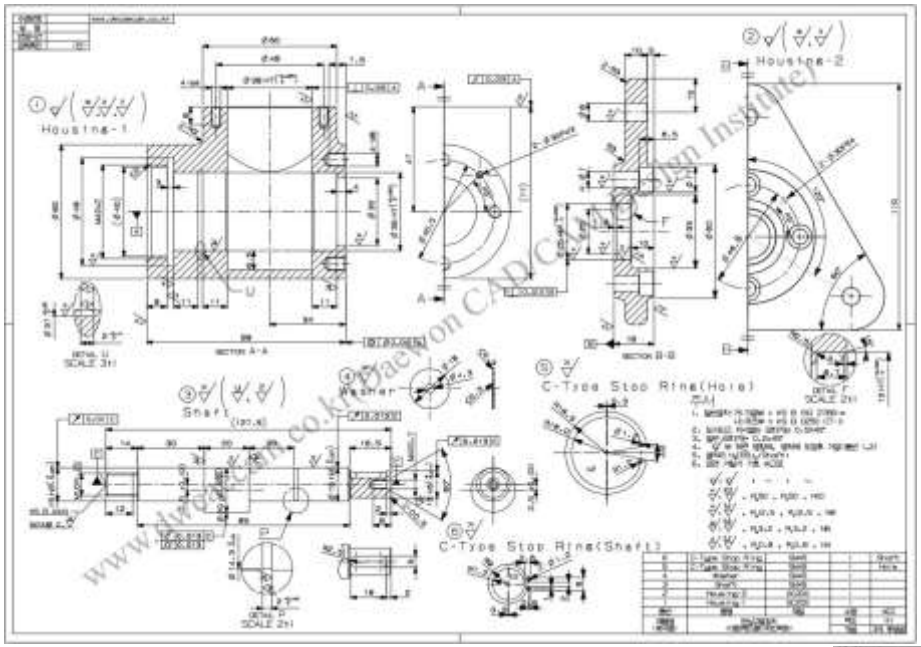


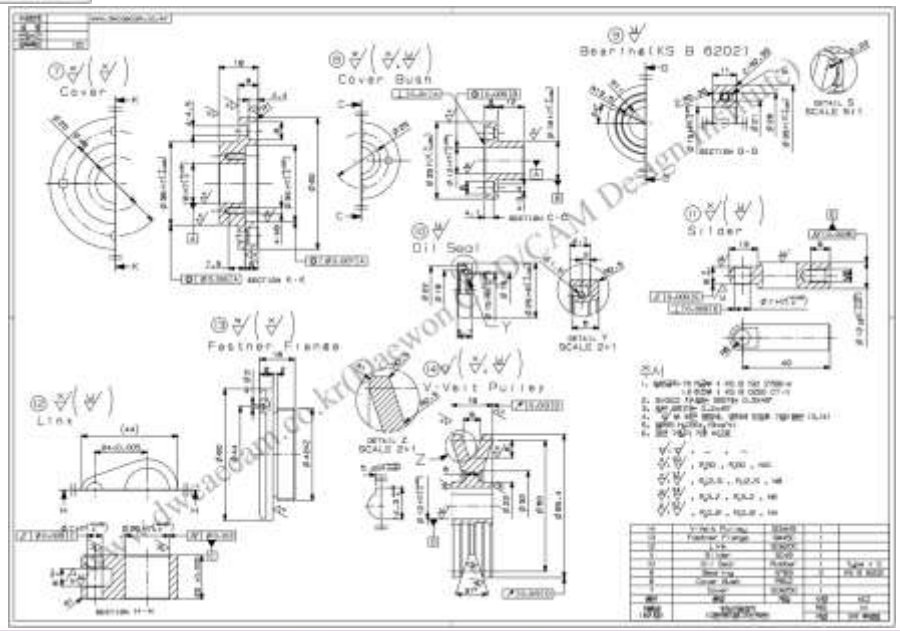

대원캐드캠디자인학원 자료


대원캐드캠디자인학원 자료


SECTION A-A
Housing Pin
CP(Connector Pin)-Pin
Fastner Frame
Housing-1
M4-Hexagonal Hole Bolt
M3-Hexagonal Hole Bolt
Slider
Cover Bush
Cover
Housing-2
M6-Hexagonal Hole Bolt
V-Vait Pulley
Washer
Key
Connector Pin
M10_Nut
Bearing
Shaft
Oil Seal
Link
C-Type Stop Ring(Hole)
C-Type Stop Ring(Shaft)
M4-Hexagonal Bolt
Bush(1)
Bush(2)




Housing-1
Housing-2
Shaft
Washer
C-Type Stop Ring(Hole)
C-Type Stop Ring(Shaft)


Cover
Cover Bush
Bearing(KS B 6202)
Oil Seal
Slider
Fastner Frame
Link
V-Belt Pulley

11. 맺음말

- NX의 달인(達人)이 되려면 NX(UG) Tool 하나만의 공부를 남은 생애 동안 적금이나 보험 든다는 생각을 가지고 자신의 IDEA를 표현하기 위한 작업 시 주변기기들의 조작을 내 손처럼 움직이게 하라.

- NX의 Expert가 되려면 자신의 생존을 위한 Tool 이기 보다는 <주마가편:走馬加鞭>의 채찍처럼 부족한 자신을 단련하는 수단의 Tool로써 하루 평균 3시간 이상으로 10년은 투자하라.

- NX의 Veteran이 되려면 Tool을 접한 시간에 비례해 자신의 미래가 지금보다 Up Grade 될 것 같은 희망이 있어서 좋고, 부족하던 영어공부를 더해서 기쁘고, 알아내는 진정한 즐거움으로 NX를 대하라.

- NX의 Master가 되려면 거북이 같은 의지, 토끼같이 재치 있는 Smart 함과, 사람이 만든 Tool 일진데 시간이 문제지 나도 하면 끝이 있겠지 라고 자신을 믿어주며 위안하는 무모한 용기를 가져보라.

- NX의 Guru(지도자)가 되려면 달력의 적색 날짜를 보며 휴식하려 하기보다 NX를 집중적으로 파고들 절호의 기회라 여기며 사명감을 가지고 산다면 가던 길을 가라! NO라면 자신과 제자를 위해 떠나라.

- 嘴琢同時(취탁동시): 가르치는 사람의 사랑, 능력과 열정이 있고 배우는 사람의 열의와 지혜가 일치할 때 교육은 성공 할 수 있다고 합니다.

질의 및 응답(Q&A)