

# STEP 표준을 이용한 제품설계-제조 연계 방안

2001. 2. 16

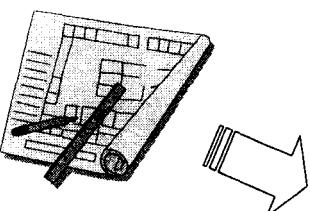
강 무진



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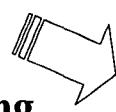
## 제품 기술 정보의 흐름



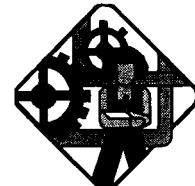
Design



Process Planning



Manufacture

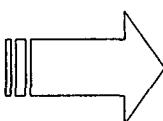
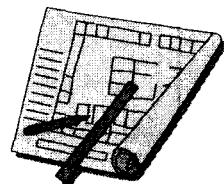


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## 공정 계획

### □ 설계와 제조의 가교(Bridge)



#### Design

- Geometry
- Function
- Cost
- etc.

#### Process Planning

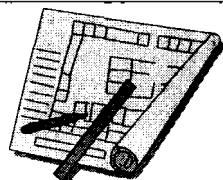
- Interpret design
- Identify machining feature
- Select process, machine, etc.
- and so on



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## 설계와 공정계획의 연계 조건



#### Design

- Geometry
- Function
- Cost
- etc.

Geometry  
Feature  
Tolerance

#### Process Planning

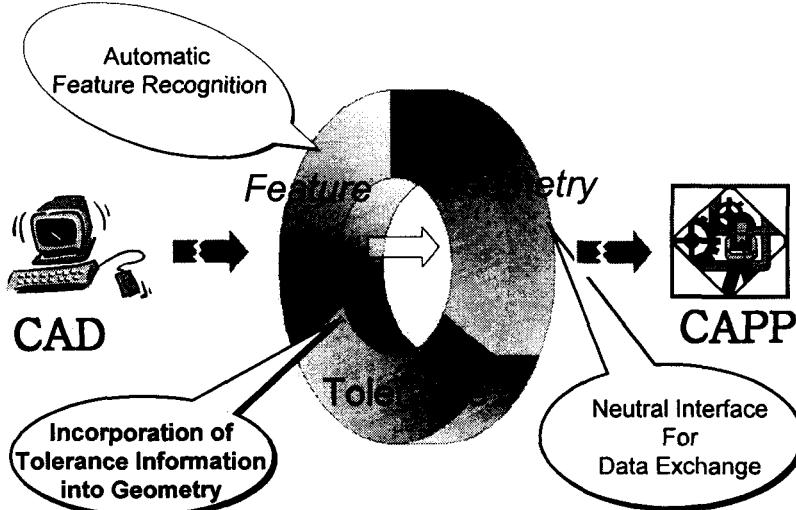
- Interpret design
- Recognize feature
- Select processes
- Determine m/c, tools
- etc.



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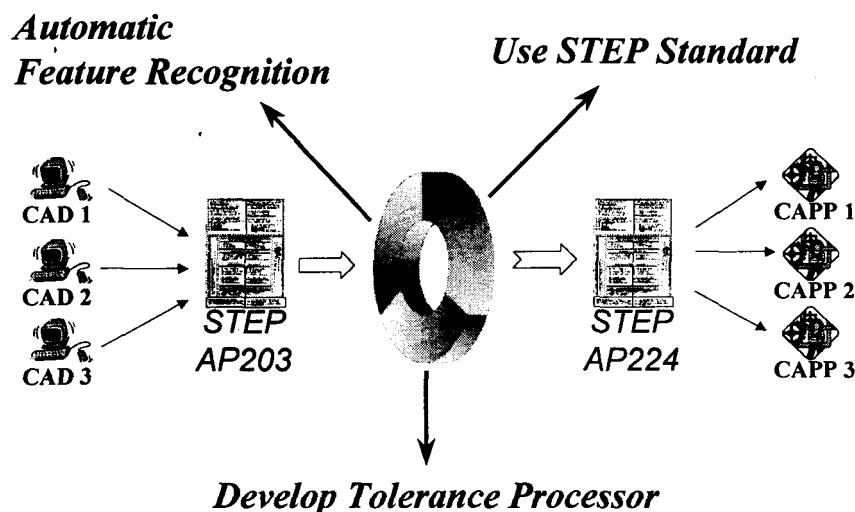
## 설계/공정계획 연계시 문제점



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## 해결 방안

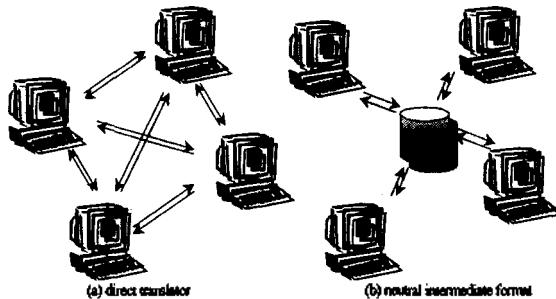


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## 제품 정보 모델 표준 - STEP

- STandard for the Exchange of Product model data
- Geometry + Feature + other Manufacturing Information

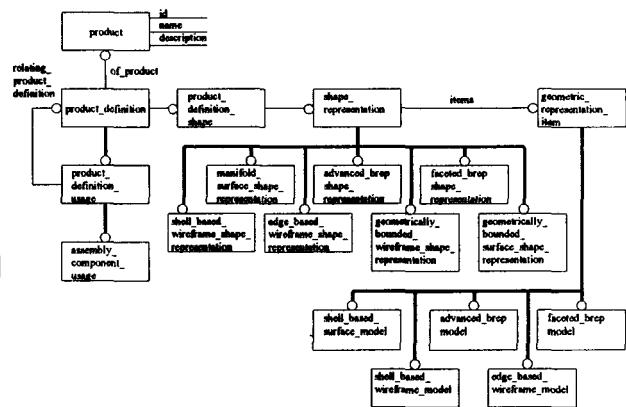


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## STEP AP 203

- Configuration controlled design
- Mainly Brep geometry information
- Documentation of design change and its process

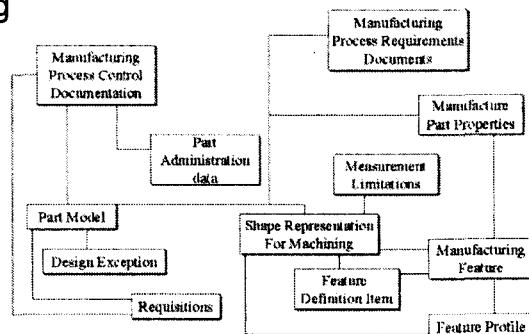


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## STEP AP 224

- Mechanical product definition for process planning using machining features
- Covering milling and turning processes
- Order information
- Requisition of raw stock
- And additional part information

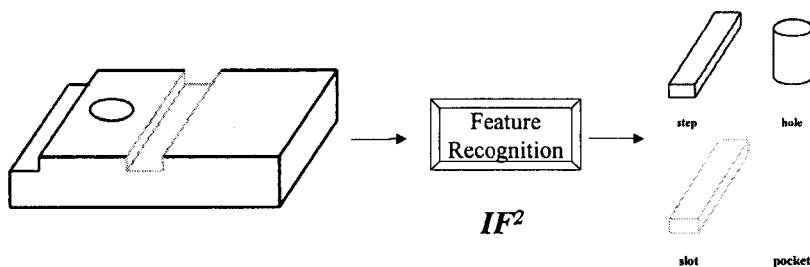


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## Machining Feature

- Representation of geometry in manufacturing terms  
(ex.) hole, pocket, slot, step, etc.

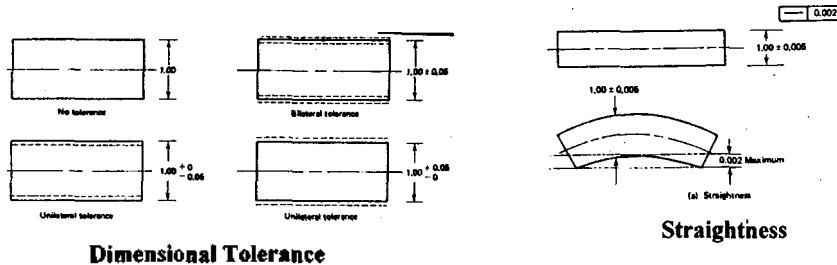


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# Tolerance Information

- Dimensional tolerance
- Surface roughness
- Geometric tolerance  
(ex.) straightness, parallelism, roundness, etc.



Dimensional Tolerance

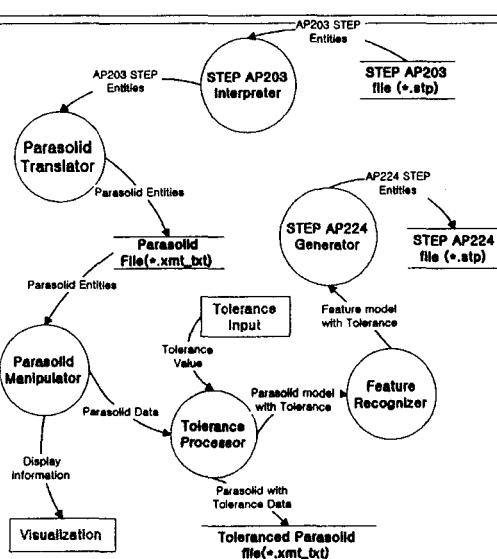
Straightness



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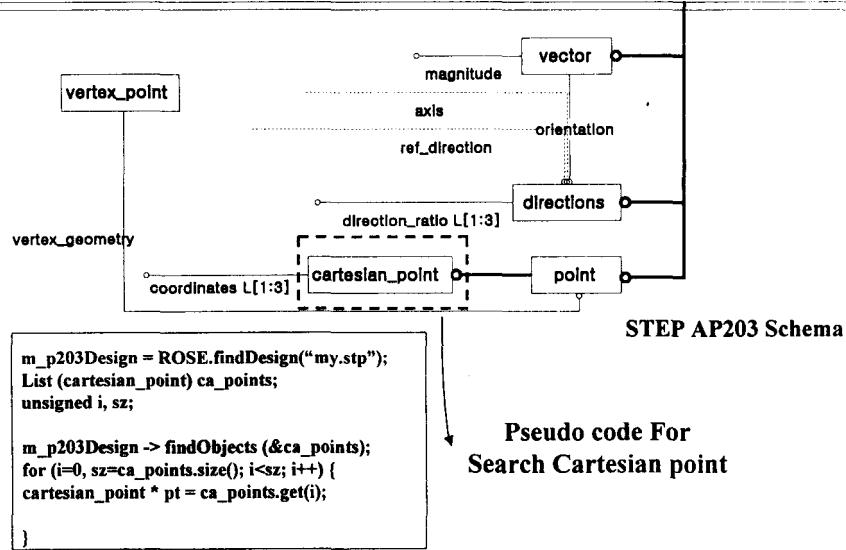
## 설계 - 공정계획 연계 흐름도



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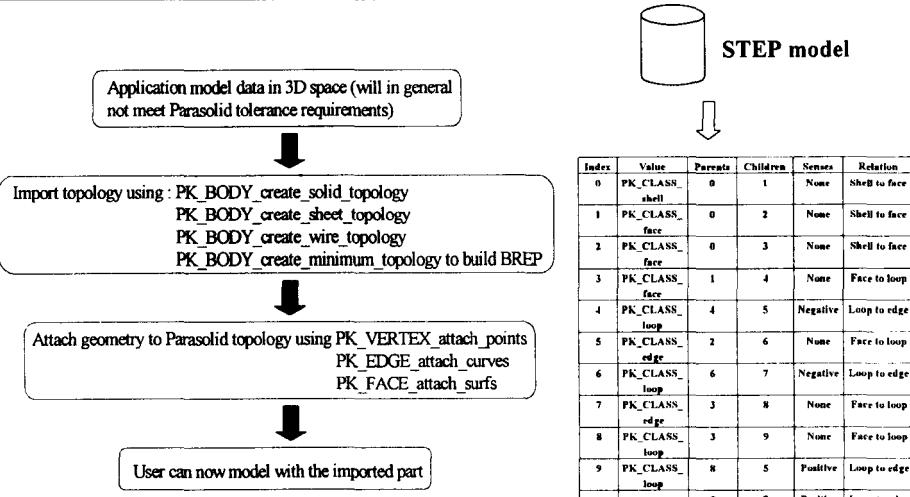
## STEP 203 해독



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## Parasolid 변환



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## 공차 처리

### □ Assign tolerance information to geometric entity

(Ex.) Linear Dimensional Tolerance

Topo. Entity	1	2	3	4			
Select Entity.	0	1	2	1			
	F1	E1	V1	E2			
Origin Entity	0	1	2	1			
Origin Entity ID	F1	E1	V1	E2			
Target Entity							
Target Entity ID							
Dimension							
Plus Tolerance							
Minus Tolerance							

```
Typedef struct {  
    Origin Entity;  
    Origin Entity ID;  
    Target Entity;  
    Target Entity ID;  
    Dimension;  
    Plus Tolerance;  
    Minus Tolerance;  
}
```

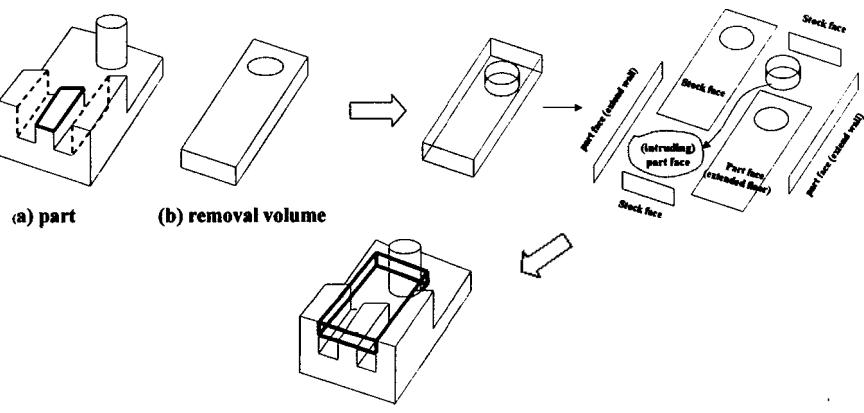


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## 가공 특징 형상 인식

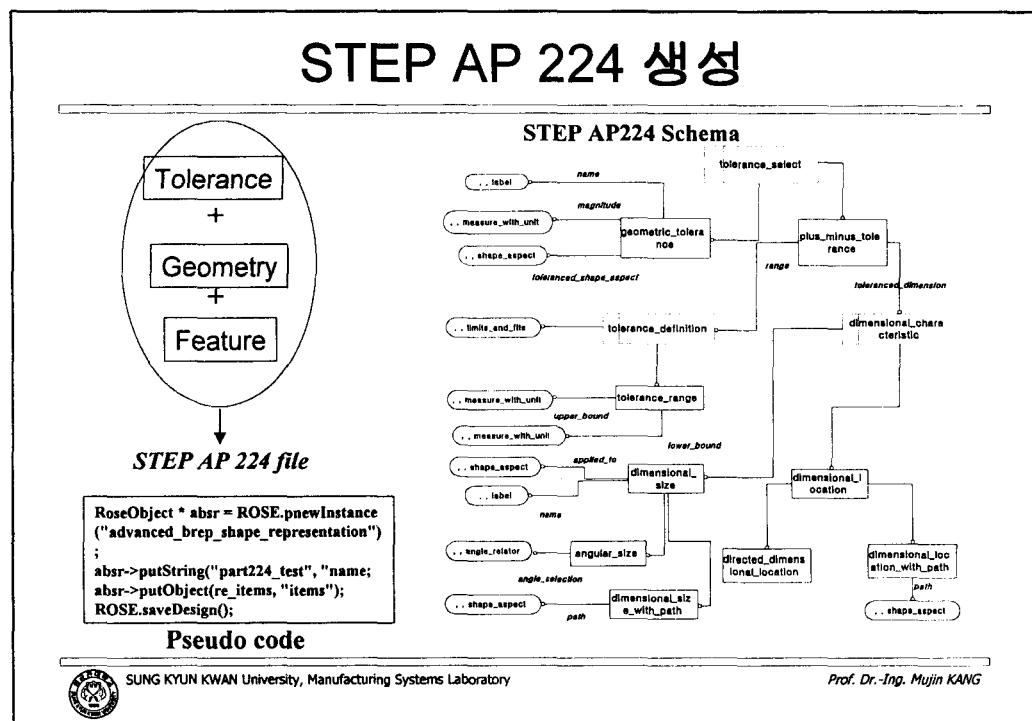
### IF<sup>2</sup>(Integrated Incremental Feature Finder)



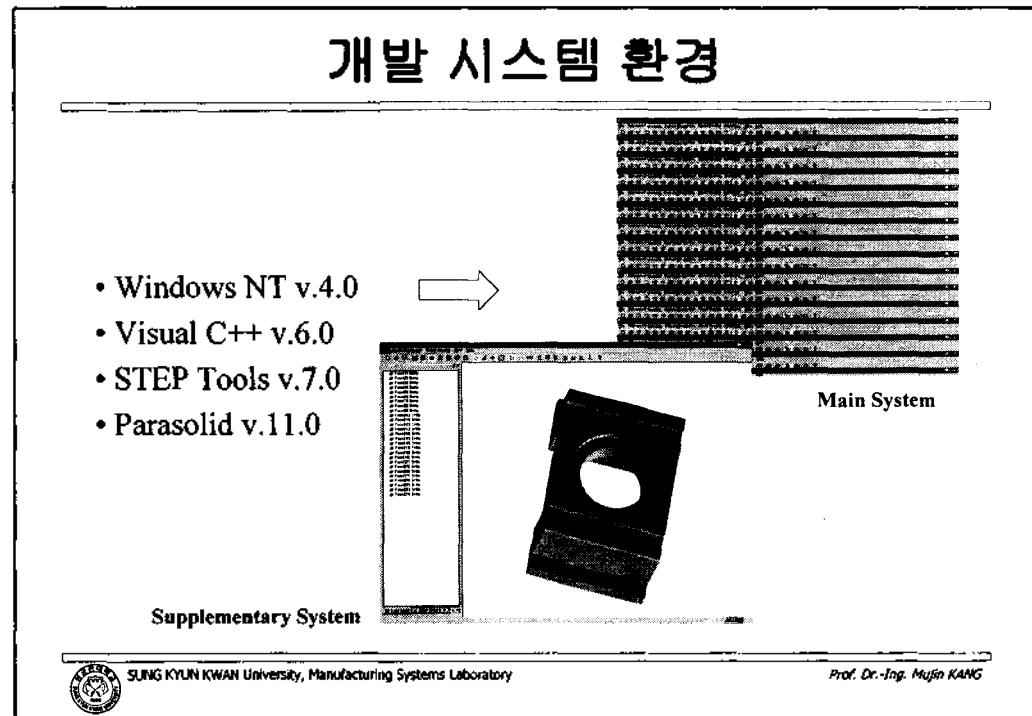
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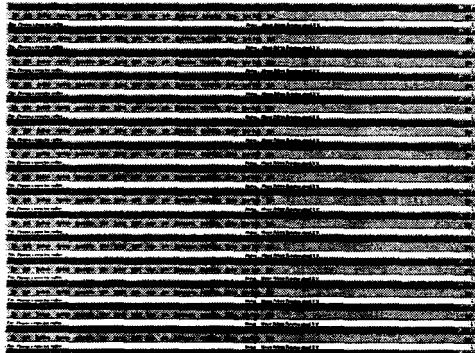
## STEP AP 224 생성



## 개발 시스템 환경



## 적용 사례 (AP203 해독)



Part modeling in the UniGraphics

```

FILE_DESCRIPTION(
/* description */(),
/* implementation_level */2,
1);

FILE_NAME(
/* name */"thesis_part",
/* time_stamp */"1999-05-28T08:25:35+09:00",
/* author */""),
/* organization */(),
/* preprocessor_version */"ST-DEVELOPER v1.5",
/* engineering_system */"EDS - UNIGRAPHICS 14.0",
/* authorization */");

3680 = AXIS2_PLACEMENT_3D(*#3650,#3660,#3670),
#3690 = PLANE(*#3680),
#3700 = ADVANCED_FACE(*#3580,#3540),#3690,T),
#3710 = ORIENTED_EDGE(*#3710,T),
#3720 = ORIENTED_EDGE(*#3720,T),
#3730 = ORIENTED_EDGE(*#3730,T),
#3740 = ORIENTED_EDGE(*#3740,F),
#3750 = EDGE_LOOP(*#3710,#3720,#3730,#3740),
#3760 = FACE_BOUND(*#3750,T),
#3770 = CARTESIAN_POINT(*#3760,D),
#3780 = DIRECTION(*#3770),
#3790 = DIRECTION(*#3780),
#3800 = AXIS2_PLACEMENT_3D(*#3770,#3780,#3790),
#3810 = PLANE(*#3800);

```

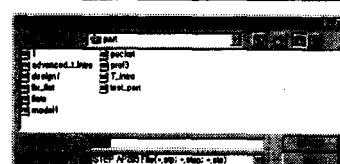
### STEP AP203 Physical file



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## 적용 사례 (STEP/Parasolid 변환)



Parasolid Physical file

```

FILE_DESCRIPTION(
/* description */(),
/* implementation_level */2,
1);

FILE_NAME(
/* name */"thesis_part",
/* time_stamp */"1999-05-28T08:25:35+09:00",
/* author */""),
/* organization */(),
/* preprocessor_version */"ST-DEVELOPER v1.5",
/* engineering_system */"EDS - UNIGRAPHICS 14.0",
/* authorization */");

3680 = AXIS2_PLACEMENT_3D(*#3650,#3660,#3670),
#3690 = PLANE(*#3680),
#3700 = ADVANCED_FACE(*#3580,#3540),#3690,T),
#3710 = ORIENTED_EDGE(*#3710,T),
#3720 = ORIENTED_EDGE(*#3720,T),
#3730 = ORIENTED_EDGE(*#3730,T),
#3740 = ORIENTED_EDGE(*#3740,F),
#3750 = EDGE_LOOP(*#3710,#3720,#3730,#3740),
#3760 = FACE_BOUND(*#3750,T),
#3770 = CARTESIAN_POINT(*#3760,D),
#3780 = DIRECTION(*#3770),
#3790 = DIRECTION(*#3780),
#3800 = AXIS2_PLACEMENT_3D(*#3770,#3780,#3790),
#3810 = PLANE(*#3800);

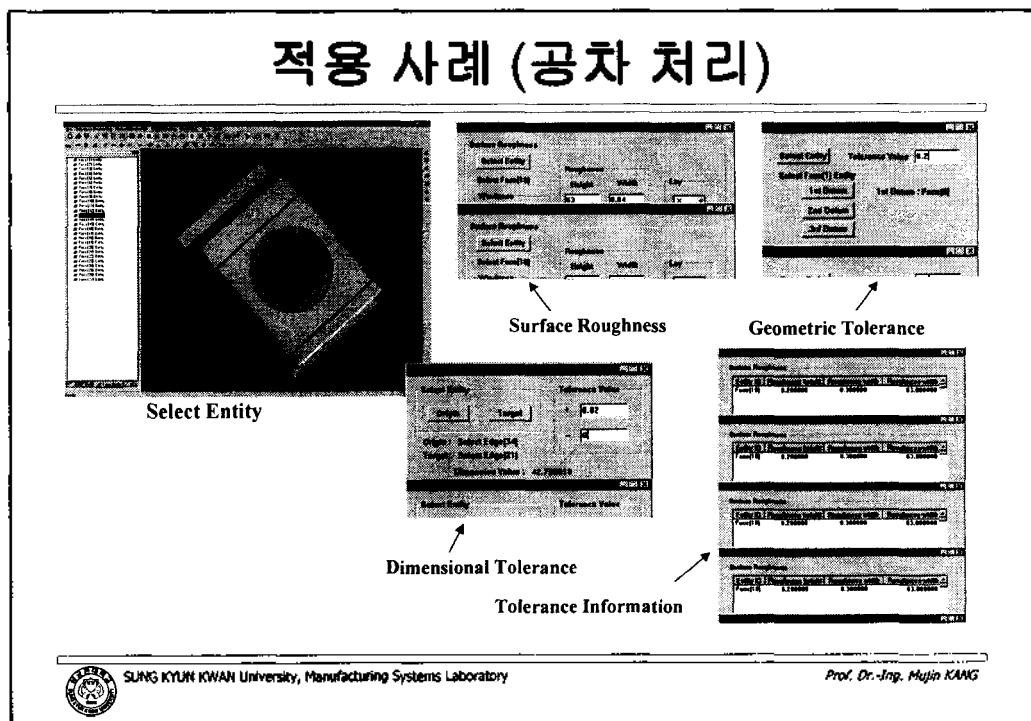
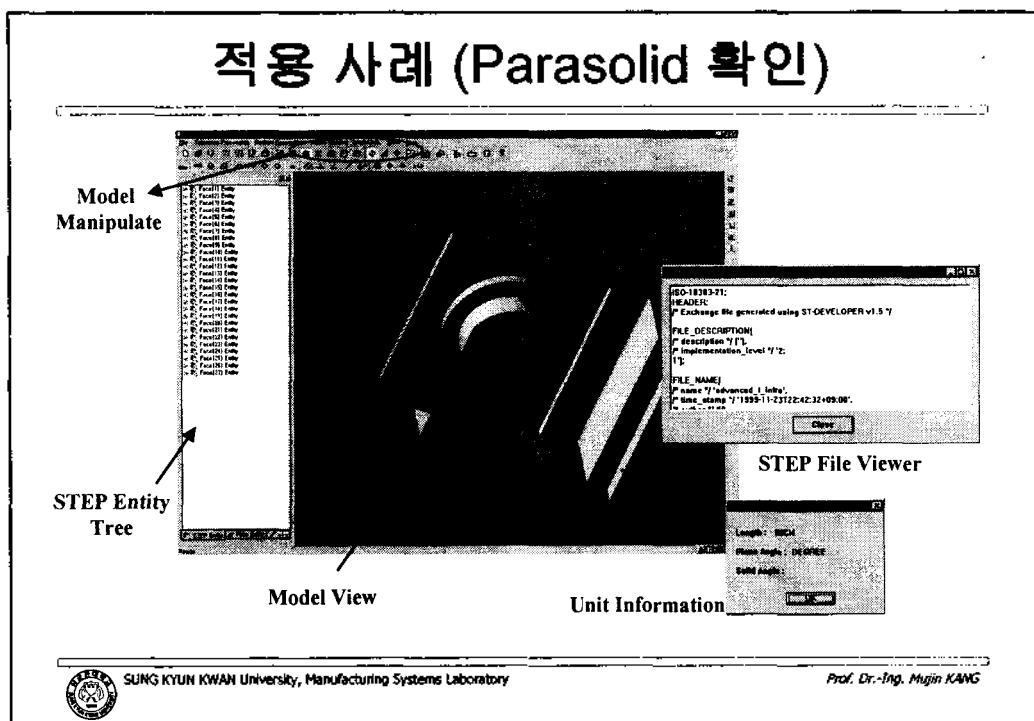

```

### STEP AP203 Physical file



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## 적용 사례 (가공 특징형상 추출)

IF<sup>2</sup> System

Select Recognized Feature

The screenshot shows the IF<sup>2</sup> System interface. On the left, there is a window titled "Feature Tree" containing a list of features: set slot(1), set slot(2), set slot(3), set slot(4), set slot(5), set pocket(6), set pocket(7), set pocket(8), set slot(9), and set pocket(10). An arrow points from this window to a larger window on the right titled "Select Recognized Feature". This window lists various recognized features and their properties, such as shape, aspect, product definition, property definition representation, and geometric tolerance.

Feature Tree

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## 적용 사례 (AP204 생성)

Create

**Export Dialog for STEP AP224**

The screenshot shows the Export Dialog for STEP AP224. It compares two versions of the dialog, likely showing the transformation process. Below the dialog, a thought bubble contains the text "Geometry + Tolerances + Features". To the right, a large block of text represents the content of the STEP AP224 Physical File, which includes detailed definitions of geometry, tolerances, and features using STEP 224 syntax.

```

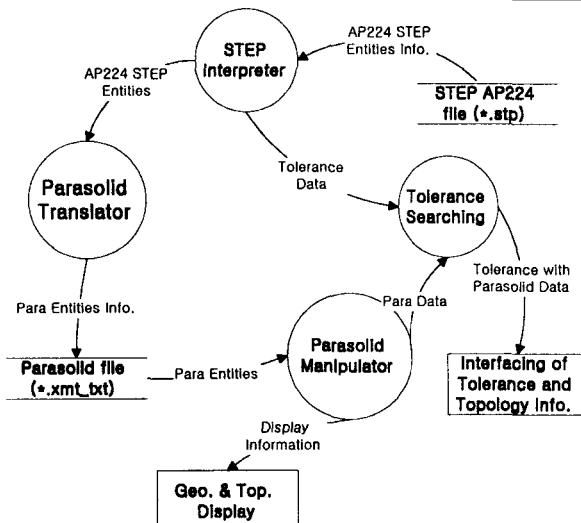
AP1-MEASURE_WITH_UNIT(PARAMETER_VALUE@00999999;
9774802000;5);
AP2-SHAPE_ASPECT(S,AS16,F);
AP3-PRODUCT_DEFINITION_SHAPE(S,5,S);
AP4-PROPERTY_DEFINITION_REPRESENTATION(WB16,W18);
AP5-REPRESENTATIONS(WB16,S);
AP6-GEOMETRIC_TOLERANCE(WB16,'Straightness',W21);
AP7-MEASURE_WITH_UNIT(PARAMETER_VALUE@00999999;
9774802000;5);
AP8-SHAPE_ASPECT(S,AS12,F);
AP9-PRODUCT_DEFINITION_SHAPE(S,5,S);
AP10-PROPERTY_DEFINITION_REPRESENTATION(WB12,W34);
AP11-REPRESENTATIONS(WB12,S);
AP12-GEOMETRIC_TOLERANCE(WB12,'Circularity',W23);
AP13-MEASURE_WITH_UNIT(PARAMETER_VALUE@01099999;
9774802000;5);
AP14-SHAPE_ASPECT(S,AS28,F);
AP15-PRODUCT_DEFINITION_SHAPE(S,5,S);
AP16-PROPERTY_DEFINITION_REPRESENTATION(WB28,W30);
AP17-REPRESENTATIONS(WB28,S);
AP18-GEOMETRIC_TOLERANCE(WB28,'Circularity',W23);
AP19-MEASURE_WITH_UNIT(PARAMETER_VALUE@005000000;
9774802000;5);
AP20-SHAPE_ASPECT(S,AS04,F);
AP21-PRODUCT_DEFINITION_SHAPE(S,5,S);
AP22-PROPERTY_DEFINITION_REPRESENTATION(WB14,W36);
AP23-REPRESENTATIONS(WB14,S);
AP24-PLUS_MINUS_TOLERANCE(WB14,W41);
AP25-TOLERANCE_RANGE(WB09,W60);

```

STEP AP224 Physical File

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## 보조 시스템 (AP204 처리기)



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## 적용 사례 (AP204 해독)

**STEP AP224 Physical File**

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### **Feature Information**

#### **Tolerance Information**

## Model Information

The seal of the University of Michigan, featuring a circular design with the text "THE UNIVERSITY OF MICHIGAN" around the perimeter and a central emblem.

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## 결론

- Representing tolerance information together with feature and geometry information in neutral format (using STEP AP 203 and AP 224)
- Preprocessor of a CAPP System
- System-independent Interfacing of CAD and CAPP



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## 향후 과제

- Covering complete set of manufacturing features
- Incorporating more efficient feature recognizer
- Processing manufacturing information in STEP model



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