Engineering Knowledge Representation

What is Knowledge?

- Data – extant facts (e.g., list of coordinates)
- Information – set of facts and relationships between them (e.g., wing profile given by list of coordinates)
- Knowledge – information with meaning attached to it – including generalizations, rules, methods, procedures (e.g., procedure for defining a wing profile)

  E.g. “The tree is green”.
  - Data: tree, green.
  - Information: tree color attribute green.
  - Knowledge: what you picture: green leaves, pine tree, green bark.

Dixon (1995), Summers (2002)

Types of Engineering Information and Knowledge

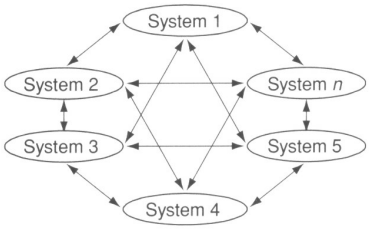
- Chan & Finger (1998): There are two types of design information: artifact and process.
- Douglas, et al. (2002): “Knowledge can be classified as declarative, procedural and strategic”.

Types of CAD data (Lee, 2000)

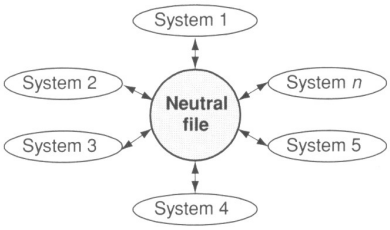
- Product Data
  - *All data arising in the product life-cycle, including design, manufacturing, QA, testing and support.*
  - Product Definition Data
    - *Artifact data.*
    - CAD drawing and associated annotation
    - solid model and associated annotation

CAD Data Exchange

Exchanging Data between Systems



Conventional Method



Neutral File Method

Neutral File Method requires fewer translators

Data Translation Problems

1. Different representation of same data
   e.g. Circular_Arc_Rep1: x_c, y_c, R, θ_1, θ_2
   Circular_Arc_Rep2: x_c, y_c, x_1, y_1, x_2, x_2

2. Incomplete Mappings

3. Numerical errors (e.g. from truncation)

DXF format

This format is for Product Definition Data, for CAD Drawing Data.

- Introduced for users of AutoCAD, it is a text (ASCII) file that can be read and modified in a text editor.
- Because of the popularity of AutoCAD it has become a de-facto standard.

Initial Graphics Exchange Specification (IGES)

This format is for Product Definition Data, for Solid Model data.

- It was developed in 1979 by a Committee of large companies, under contract from the U.S. Air Force.
- It can be in ASCII or binary format.

STandard for the Exchange of Product data (STEP)

This format is for all Product Data.

- The Product Data Exchange Standard (PDES) was started in the U.S. in 1983.
- STEP was started by the International Organization for Standardization (ISO) in 1984
- PDES joined STEP in 1985.
- STEP is expected to become the primary standard for the U.S. armed forces and U.S. companies.

STandard for the Exchange of Product data (STEP)

- Parts 203 and 214 of STEP (ISO-10303) define entities for solid model geometry.
- The STEP file format looks like:

File format defined in Part 21 of STEP.

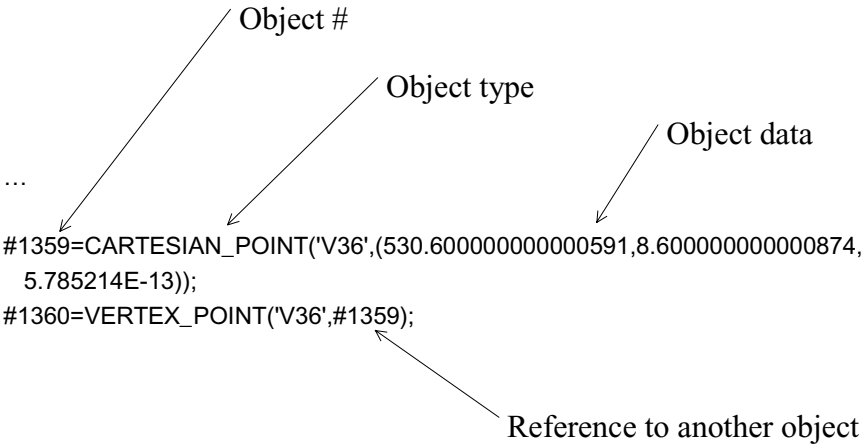
```
ISO-10303-21;
HEADER;
FILE_DESCRIPTION((''),'2;1');
FILE_NAME('/home/mefac/bettig/sdrc/dogbone.stp','2001-12-11T21:04:35',('User'),('SDRC'),'I-DEAS
    Master Series 8','UNIX','Yes');
FILE_SCHEMA(('AUTOMOTIVE_DESIGN { 1 2 10303 214 -1 1 5 3 }'));
ENDSEC;
DATA;
#10=(NAMED_UNIT(*)PLANE_ANGLE_UNIT()SI_UNIT($,.RADIAN.));
#11=DIMENSIONAL_EXPONENTS(0.0,0.0,0.0,0.0,0.0,0.0,0.0);

...
```

Header info. start

Product info. start

STEP Part 21 format



The EXPRESS language

- The entities in STEP (e.g. cartesian_point) are defined using the EXPRESS language.

  E.g.:

```
ENTITY cartesian_point
  SUBTYPE OF (point);
  coordinates  : LIST [1:3] OF length_measure;
END_ENTITY;
```

- Express has an object-oriented flavor (it has inheritance).
- Express is defined in Part 11 of STEP
- What other information languages are there?

The EXPRESS Language

- Words of the EXPRESS language include:
  - SCHEMA, REFERENCE FROM
  - REAL, INTEGER, BOOLEAN, ENUMERATION OF, SET, LIST
  - TYPE, END_TYPE (like typedef in C++)
  - ENTITY, END_ENTITY (~like class in C++)
  - SUBTYPE OF, SUPERTYPE OF
  - WHERE (for condition statements)
  - FUNCTION

STEP Geometry

- Part 42 of STEP gives the core geometry schema

– see www.mel.nist.gov/sc5/soap/

- Other parts of STEP give schemas for Drafting Data, Finite Element Models, and application specific schemas such as for ship piping, layered electrical products and composite structures.