Data Exchange between CAD/CAE/CAM Systems

Description of product or product definition data:

- Engineering Drawings (Manual or Electronic):
  - Vector data for lines (solid lines, dotted lines, centre lines, dimension lines, and extension lines)
  - Annotation data for the dimension values, notes, and symbols in the drawing.
- CAD Systems:
 - A solid model representation with some associated annotation data.

Driving Forces for CAD/CAM Data Exchange

- Fundamental incompatibilities among entity representations
- Complexity of CAD/CAM systems
- The varying requirements of users
- Restrictions on access to proprietary database information
- Rapid pace of technological change

Requirements for the Exchange

- Shape data: both <u>geometric</u> and <u>topological</u> information, part or form features. Fonts, color, annotation are considered part of the geometric information.

- Non-shape data: <u>graphics</u> data such as shaded images, and <u>model global</u> data as measuring units of the database and the resolution of storing the database numerical values.

- Design data: information that <u>designers</u> generate from geometric models for <u>analysis</u> purposes. Mass property and finite element mesh data belong to this type of data.

- Manufacturing data: information as tooling, NC tool paths, tolerancing, process planning, tool design, and bill of materials (BOM).

Exchange Methods
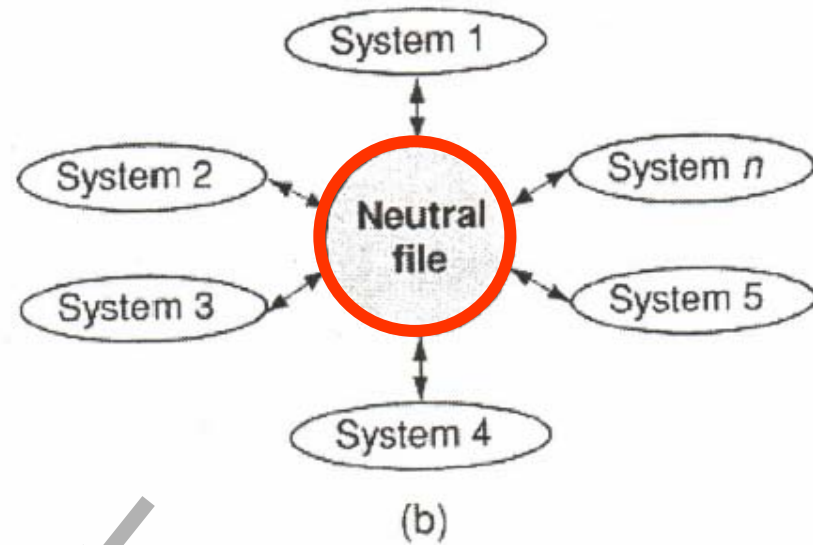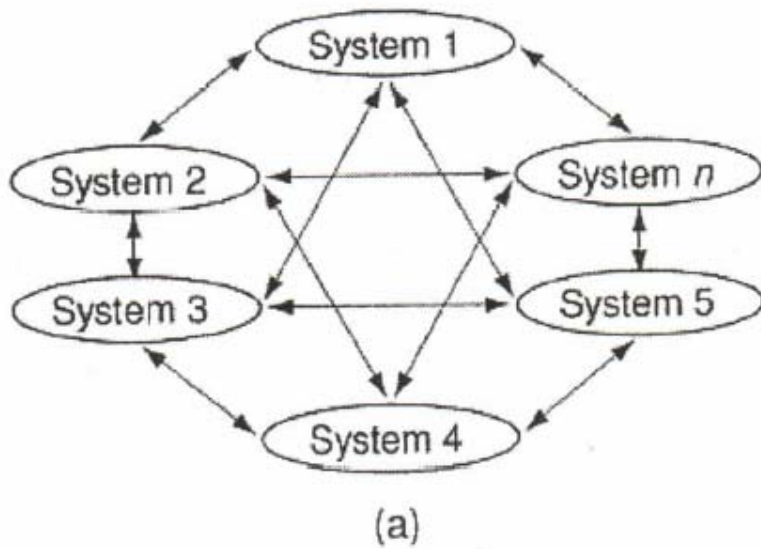
Standard neutral data formats:

- Initial Graphics Exchange Specification (**IGES**) - the most popular format of the neutral file, supported by all CAD/CAE/CAM systems and defined by the international standard organization (ISO).

- Drawing Interchange Format (**DXF**) - a format originated by AutoDesk and used mainly for the exchange of drawing data.

- Standard for The Exchange of Product Model Data (**STEP**) - the standard data format used to store all the data relevant to the entire life cycle of a product, including design, analysis, manufacturing, quality assurance, testing, and maintenance, in addition to the simple product definition data. The data format was also called **PDES** (Product Design Exchange Specification) at the early stage of its development in North America.

A number of other neutral data formats for CAD/CAE/CAM systems were used in the past. These include PHIGS, NAPLPS and GKS.

Currently, CAD systems, which used to support IGES format, are moving toward the use of STEP.

IGES (Initial Graphics Exchange Specification)

- first developed by <u>National Institute of Standards and Technology</u> (NIST) in 1980.

- then adopted by the <u>American National Standards Institute</u> (ANSI) in the same year.

- exchanges primarily shape (both geometric and topological) and non-shape data, which is referred as CAD-to-CAD exchange

IGES (Initial Graphics Exchange Specification)

- codes a superset of common entities of all CAD/CAM systems to facilitate the translation between various systems
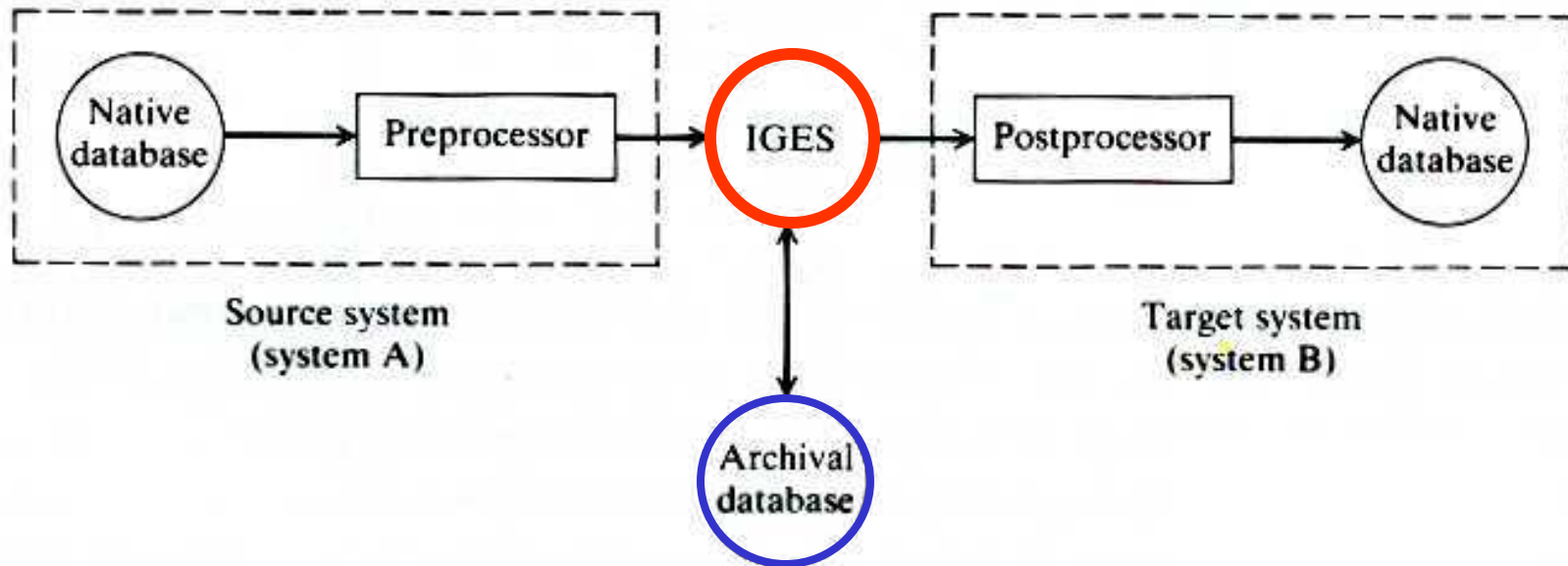
Development of IGES

As a result of this effort, IGES, version 1.0, was published in January 1980. It became an American National Standards Institute (ANSI) standard in September 1981. IGES was the first standard exchange format developed to address the need to communicate product definition data between different CAD/CAM/CAE systems. Additional capabilities were added later.

- IGES version 1.0 - mainly addresses the exchange of CAD drawings.
- IGES version 2.0 - supports the exchange of finite-element data and printed circuit-board data
- IGES version 3.0 - enhances the capabilities of user-defined Macros that are essential to exchange the standard part libraries.
- IGES version 4.0 - supports the CSG tree of a solid,
- IGES version 5.0 - handles the B-rep data of solids.

IGES Format

An IGES file is composed of six sections in the following order. A record is a line comprising 80 characters.

Originally based on FORTRAN data format

1. Flag (optional),
2. Start,
3. Global,
4. Directory Entry (DE),
5. Parameter Data (PD), and
6. Terminate

In IGES File

The Flag section is used only with the compressed ASCII and binary formats. IGES data in a file can be represented in two formats: ASCII and binary. The ASCII format comprises two types: a fixed 80-character record length and a compressed form. The compressed form is simply an ASCII file compressed by eliminating spaces from the records.

The Start section provides a human-readable description of the file, such as the sending system that generated the original data, the pre-processor, and the product being described.

The Global section includes information describing the pre-processor and information needed by the post-processor to interpret the file. Some of the items specified in this section are

- the characters used as delimiters between individual entries and between records
- the name of the IGES file itself,
- the number of significant digits in the representation of integers and floating point numbers on the sending system,
- the date and time of the file generation,
- the model space scale,
- the model units,
- the minimum resolution and maximum coordinate values, and
- the name of the author of the file and organization.

In IGES File

The Directory Entry section is a list of all the entities together with certain of their attributes. In an IGES file, all product definition data are expressed as a list of predefined entities - the geometric entities such as lines, curves, planes, and surfaces and the annotation entities such as notes and dimension values. Each entity is assigned a specific entity type number. The entry for each entity occupies two 80-character records that are divided into a total of twenty 8-character fields. The first and the eleventh (beginning of the second record of any given entity) fields contain the entity type number. The second field contains a pointer to the record in the Parameter Data section where the actual data defining the entity are stored. This pointer is simply the sequence number of the relevant record in the PD section. The remaining fields are used to store the attributes, such as line fonts, layer number, transformation matrix, line weight, and color.

The Parameter Data section contains the actual data defining each entity listed in the Data Entry section. For example, a straight-line, entity is defined by the six coordinates of its two end points. While each entity always has two records in the DE section, the number of records needed for each entity in the PD section varies from one entity to another and depends on the amount of necessary data. Parameter data are placed in free format in columns 1 through 64. The parameter delimiter specified in the Global section is used to separate parameters, and the record delimiter - also specified in the Global section - is used to terminate the list of parameters. Usually, commas are used as parameter delimiters, and semicolons are used as record delimiters. Columns 66 through 72 in all PD records contain the pointer to point back to the corresponding entity in the DE section. Note that this pointer has the sequence number of the first of the two records of each entity in the DE section. Columns 74 through 80 contain the sequence number of its own record. The Terminate section contains a single record that specifies the number of records in each of the four preceding sections for checking purposes.
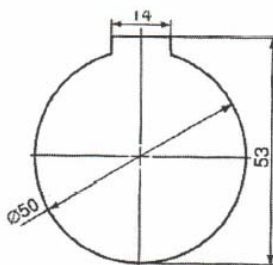
Example IGES File

IGES file generated from an AutoCAD drawing by the IGES translator from Autodesk, Inc., translator version IGESOUT-3.04.

An Example IGES File for the 2D Keyed Hole.

DRAWING INTERCHANGE FORMAT (DXF)

Drawing interchange format (DXF) files were originally developed to give users flexibility in managing data and translating AutoCAD' drawings into file formats that could be read and used by other CAD/CAM/CAE systems.

Because of the popularity of AutoCAD, DXF became the de facto standard of interchanging CAD drawing files for almost all CAD/CAM/CAE systems. In fact, almost every newly introduced CAD/CAM/CAE system tends to provide translators to and from the DXF file.

A DXF file is an ASCII text file and consists of five sections:

- **Header** - describes the AutoCAD drawing environment that existed when the DXF file was created.

- **Table** - contains information about line types, layers, text styles, and views that may have been defined in the drawing.

- **Block** - contains a list of graphic entities that are defined as a group.

- **Entity** - immediately follows the Block section, and serves as the main part of the DXF file, with all entities of the drawing described in it.

- **Terminate** - indicates the end of the file.

Similar to that of the IGES files, the repertoire of the entities used in DXF files has been expanded with the introduction of the newer versions of AutoCAD.

A DXF file created by a higher version of AutoCAD cannot be read by other systems based on a lower version of DXF.

Limitations of **IGES** and **DXF**

The IGES files and DXF files were developed to exchange product definition data instead of product data. By product data we mean the data relevant to the entire life cycle of a product (e.g., design, manufacturing, quality assurance, testing, and support).

Even though the specification of the IGES or DXF file has been broadened to encompass some of these product data, the data carried by those files are inherently insufficient to be the product data supporting the entire life cycle.

PDES (Product Data Exchange Standard)
(then Product Data Exchange Using STEP)

- to support any industrial application such as mechanical, electric, plant design, and architecture and engineering construction

- to include all four types of data which is relevant to the <u>entire life-cycle of a product</u>: design, analysis, manufacturing, quality assurance, testing, support, etc.

PDES

- PDES is a much more comprehensive and complex standard than IGES or any other predecessors
- The user interface is not as simple as "put IGES" and "get IGES."
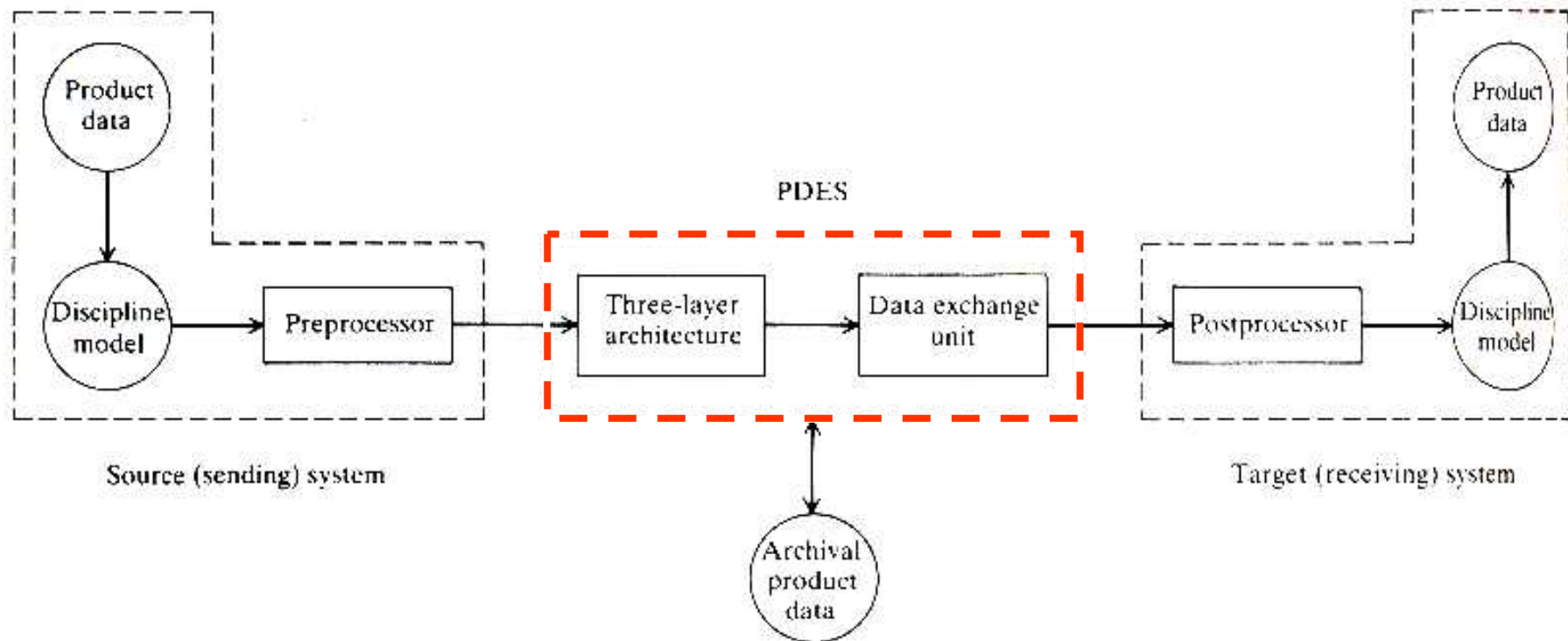
STEP (Standard for the Transfer and Exchange of Product Model Data)

- an ISO standard for the exchange of product data.

- The goal of this standard is to enable the exchange of a computerized product model with all its supporting types of data in a neutral format (all four types of data).

Introduction of **STEP**

Meanwhile, the International Organization for Standardization's (ISO's) Technical Committee TC184 (Industrial Automation Systems) and its subcommittee SC4 (External Representation of Product Model Data) were formed in July 1984 to establish a single worldwide standard for the exchange of product model data, STEP (Standard for the Exchange of Product model data).

Both PDES and STEP had the identical goal, so in June 1985 the IGES Steering Committee decided that PDES should represent U.S. interests in the STEP effort. As a result, the acronym PDES has been changed to stand for product data exchange using STEP, to emphasize the intention that **PDES and STEP be identical.**

APPLICATION PROTOCOLS AND ASSOCIATED ABSTRACT TEST SUITES

I 201 Explicit draughting (W)*

I 202 Associative draughting (C)

I 203 Configuration-controlled design (C)

C 204 Mechanical design using boundary rep (C)

C 205 Mechanical design using surface rep (W)

X 206 Mechanical design using wireframe (X)

E 207 Sheet metal die planning and design (C)

W 208 Life-cycle product change process (W)

C 209 Compos & metal struct, anal, & related dgn (W)

C 210 Electronic P-C assy: product-design data (W)

X 211 Electronic P-C assy: test, diag, & rem anuf (W)

C 212 Electrotechnical design and installation (W)

E 213 Num contr (NC) process plans for mach d parts (W)

C 214 Core data for automotive mech dgn processes (W)

W 215 Ship arrangements (W)

W 216 Ship moulded forms (W)

W 217 Ship piping (W)

W 218 Ship structures (W)

X 219 Dimension inspection (X)

O 220 Printed-circuit assemblies: mfg planning (O)

C 221 Process plant functional data & its schem rep (W)

W 222 Design-manuf for composite structures (W)

W 223 Exc of dgn & mfg product info for cast parts (W)

E 224 Mech parts def for p. plg using mach'n'g feat (W)

E 225 Structural bld-g elem using explicit shape rep (W)

W 226 Ship's mechanical systems (W)

E 227 Plant spatial configuration (W)

O 228 Building services: HVAC (O)

W 229 Forged parts (W)

W 230 Building structural frame: steelwork (W)

W 231 Process-engineering data (W)

W 232 Technical data packaging: core info & exch (W)

O   Neutral optical-data-interchange format (O)

O   Product life-cycle support—NATO (O)

O   SGML and industrial data (O)

IMPLEMENTATION METHODS

I 21 Clear text encoding of ex ch str.	C 24 Late C (binding for #22)
E 22 Standard data access interface	X 25 Late FORTRAN
E 23 Early C++ (binding for #22)	E 26 IDL (binding for #22)

CONFORMANCE TESTING METHODOLOGY FRAMEWORK

I 31 General concepts

E 32 Requirements on testing labs and clients

X 33 Abstract test suites

C 34 Abstract test methods for Part 21 impl.

W 35 Abstract test methods for Part 22 impl. (Approved for new scope)

DESCRIPTION METHODS

I 1 Overview and fund. principles (Amend. 1 = 0)

I 11 EXPRESS lang ref man. (Ed 2 = W)

X 12 EXPRESS I lang ref man (Type 2 tech report = W)

INTEGRATED-INFORMATION RESOURCES

INTEGRATED-APPLICATION RESOURCES

I	101 Draughting	C	104 Finite element analysis
X	102 Ship structures	I	105 Kinematics
X	103 E/E connectivity	W	106 Building core model
		A	107 Engineering anal core

INTEGRATED-GENERIC RESOURCES

I	41 Fund of prdct descr & spt (ed2 = A)	I	46 Visual presentation
I	42 Geom & topol rep (Amd1 = W)	F	47 Tolerances
I	43 Repres specialization (ed2 = A)	X	48 Form features
I	44 Product struct config (ed2 = A)	F	49 Process structure & properties
F	45 Materials		

APLLICATION-INTERPRETED CONSTRUCTS

C	501 Edge-based wireframe	C	511 Topol-bounded surface
C	502 Shell-based wireframe	C	512 Faceted B-representation
C	503 Geom-bounded 2D wireframe	C	513 Elementary B-rep
C	504 Draughting annotation	C	514 Advanced B-rep
C	505 Drawing structure & admin	C	515 Constructive solid geometry
C	506 Draughting elements	X	516 Mechanical-design context
C	507 Geom-bounded surface	C	517 Mech-design geom presentation
C	508 Non-manifold surface	C	518 Mech-design shaded presentation
C	509 Manifold surface	C	519 Geometric tolerances
C	510 Geom-bounded wireframe	C	520 Assoc draughting elements

ANSYS Workbench Environment

CAD Associativity and Bi-Directional Associativity of ANSYS Products

The ANSYS Workbench Environment uses a unique Plug-In architecture to maintain associativity with the CAD systems for solid and surface models, allowing you to make design changes to your CAD model without having to reapply any of the loads and or supports. Bi-directional associativity in Workbench Environment advances your productivity one step further by providing control of your key CAD model parameter in the Workbench Environment parameter manager. With feature, you can create simulations with multiple CAD models and Workbench Environment will automatically update the geometry for each simulation situation.

Below are a list supported CAD systems and file types:

Associative

- Unigraphics
- Pro/ENGINEER
- Autodesk Inventor
- Autodesk MDT
- Solid Edge
- SolidWorks

Non-Associative

- CATIA
- SAT
- Parasolid

If your company uses multiple CAD systems, multiple geometry interfaces may be used with your Workbench products. This way, when working in a multi-CAD environment, Workbench maintains the same operational look, feel and capability. You can then minimize your training effort and establish common simulation methods that will permit confident results.

Possible Information Loss

- Like any language translation, there is always information loss during the product data information translation
- Examples
  - A design tolerance is captured as a text string placed on a drawing, its meaning, as well as its numerical value, are lost.
  - Circular cylinders (a hole) is represented by NURBS; milling vs. drilling
  - Offset surfaces; the offset info is lost.
- Integrated CAD/CAE/CAM packages

Quick Questions

- What is IGES, PDES, and STEP?
- What are the limitations of IGES?
- What need to be transferred?
- Why can't the information be completed exchanged between CAD tools?