A Comparative Study of Product Data Exchange among CAD Systems

Conference Paper · May 2017

3 authors, including:

Faiz Mustafa
University of Baghdad

Ahmed Al-Ashaab
Cranfield University

A Comparative Study of Product Data Exchange among CAD Systems

Faiz F. Mustafa
Al-Khwarizmi College of Engineering
University of Baghdad
Baghdad, Iraq
Dr.faiz@kecbu.uobaghdad.edu.iq

Ahmed Al-Ashaab
Cranfield University
Bedfordshire, United Kingdom
a.al-ashaab@cranfield.ac.uk

Hussein A. Al-Amili
Al-Khwarizmi College of Engineering
University of Baghdad
Baghdad, Iraq
alamilihussein@gmail.com

***Abstract*— Computer Aided Design (CAD) systems have their own format of data; the problem is that, for each system, there is a need to interpret the data of other systems. In this paper, a proposed model was created and exchanged among CATIA, Pro/Eng., Solidworks, Solidedge and NX systems by using IGES and STEP neutral format standards, and a comparison was made between them to review and identify the effect of translation on data: like the processed entities, surface area and file size. The study also included case studies from Electronic access control systems Company in the United Kingdom. The results showed that STEP is better for solid translation than IGES while IGES is better than STEP for surfaces translation. File size of STEP is less than IGES starting from 39% to 65% decreasing percent. The study also found that NX, Solidedge and Solidworks are more flexible in dealing with imported data than Pro/Eng. and CATIA. The paper also included applying suggested tools to deal with dumb designs and found that the non-parametric method is more effective than feature recognition tool for imported dumb design modification.**

Keywords—CAD Systems; Data Exchange; Neutral Format; IGES; STEP.

I. Introduction

Many Computer Aided Design (CAD) systems are used to help with the geometric modeling. These systems vary from simple 2D drafting packages to fully integrated 3D systems. Each system has its own format of data and if the information of a specific system has to be entered into different systems leads to errors and redundancy [1]. The problems of exchanging data intent between dissimilar cad systems first became widely appreciated in 1980 [2]. These problems have been taken on greater importance, because of their effect on required time and cost to design and manufacturing. Data are usually exchanged within companies and between suppliers Original Equipment Manufacturers (OEMs), first-tier suppliers, sub-tier suppliers, and tooling suppliers [3]. One of the OEMs estimates that electronic CAD data exchange alone happens at least 7,000 times per month, this estimate does not include the transfer of CAD/CAM data. OEMs need from their suppliers to provide product data in the native format that OEMs' choose; sub-tier suppliers are often too small to maintain various platforms or translators. Thus, first-tier suppliers often incur the interoperability problem costs (but may pass these costs to the OEMs). Neutral format translation is one of the methods for exchanging data between different systems. Alternative two solutions of neutral format are used most often in the auto industry to exchange CAD data: Initial Graphics Exchange Specifications (IGES) and Data Exchange File (DXF) [1]. The shortcomings of these neutral format translators have led to the development of the ISO STEP standard [4]

II. Neutral Format Translators

Neutral formats may be defined in a loose way as a kind of CAD/CAM international language. They consist of formats, specifications and standards to which all suppliers of CAD/CAM systems have access. Each CAD/CAM system supplier has to make the ability to write to (the pre-processor) and read from (the post-processor) the neutral format [4]. Both pre and post-processors are likely to be written and supported by the CAD/CAM system supplier [5].

III. CAD Systems

The analysis program aimed at study five CAD systems which cover different marketplaces and widely used in companies and industries. Five systems were used: CATIA (proprietary kernel), Pro/Engineer (Grannite kernel), Solidworks (Parasolid kernel), NX (Parasolid kernel) and Solidedge (Parasolid kernel).

IV. Geometric Modeling Creation

Symmetric methods and steps of construction used in the creation of the model in the systems. sample model was produced in all the systems, the sample model comprises rotational and prismatic forms and contains features common to engineering applications, the same basic modeling procedure has been followed in every CAD system to avoid possible differences in the model being tested. Fig. 1, shows the tested geometry.

978-1-5386-0802-9/17/$31.00 ©2017 IEEE

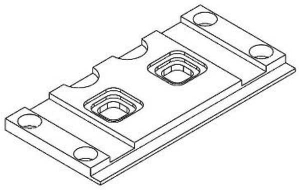


Fig. 1. The tested model.

V. CAD DATA EXCHANGE EXPERIMENTS PROCEDURE

In the data exchange experiments of this study, the tested model in each CAD system is translated to one of the neutral formats then imported to the other four systems.

VI. NEUTRAL FORMAT STANDARDS PRE-PROCESSORS

The native model was converted in each system by IGES and STEP format Pre-processor. IGES pre-processor of all CAD systems support the conversion of native models into IGES surface format. IGES preprocessor of Solidedge, Pro/Eng., and Solidworks also support the conversion of native models into IGES solid format. STEP preprocessor of all systems support the conversion of native model into STEP solid format. STEP preprocessor Solidedge also support the conversion of native models into STEP surface format.

A. File Size

STEP format results in low file size compared with IGES for the model. Neutral format pre-processor for NX model gives the lowest value of file size for the model in STEP format compared with STEP neutral format surface format that reported from Solidedge which had the highest file size value in (kB). Fig. 2, shows the file size of each format preprocessor.

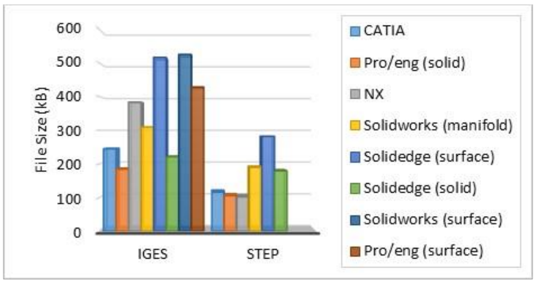


Fig. 2. IGES and STEP file size for the model.

B. The Processed Entities

The model was processed into different types of IGES and STEP entities; the results of entities after conversion are shown in table 1, these entities include points lines circles surfaces... etc. It is obvious from table 1 that the surfaces were converted into IGES trimmed parametric surfaces (type 144) except for NX the surfaces were converted into rational B-Spline surfaces (type 128). The differences between the neutral format pre-processors circles results and the real number of them can be noticed and it shows that STEP results are near the real number (76 curves).

TABLE 1. IGES AND STEP PRE-PROCESSOR RESULTS FOR THE MODEL.

IGES									
Entity	***CATIA***	***Pro/Eng.***		***NX***	***Solidworks***			***Solidedge***	
Type	***IGES***	***IGES-Surface***	***IGES-Solid***	***IGES***	***trimmed***	***manifold***	***bounded***	***Surface***	***Solid***
100	144	120	68	150	166	166	150	120	60
102	98	192	-	104	192	-	-	184	-
108	32	-	-	32	-	-	-	-	-
110	384	640	222	212	417	417	397	406	146
116	-	-	-	24	-	46	-	-	76
120	36	34	34	-	34	34	30	30	-
122	-	-	-	-	-	-	-	46	-
124	144	120	68	10	150	150	134	120	60
126	-	68	-	292	444	-	-	112	-
128	-	10	46	76	46	-	46	-	-
141	-	-	-	-	-	-	92	-	-
142	98	96	-	60	96	-	-	92	-
143	-	-	-	-	-	-	76	-	-
144	82	80	-	44	80	-	-	76	-
186	-	-	1	-	-	1	-	-	1
190	-	-	-	-	-	46	10	-	46
192	-	-	-	-	-	-	-	-	30
314	-	8	8	3	1	-	1	76	76
402	-	-	-	-	14	14	14	-	-
410	-	-	-	8	-	-	-	-	-
502	-	-	1	-	-	-	-	-	-
504	-	-	1	-	-	-	-	-	-
508	-	-	96	-	-	-	-	-	-
510	-	-	80	-	-	-	-	-	-
514	-	-	1	-	-	1	-	-	1

STEP						
Entities	***CATIA***	***Pro/Eng.***	***NX***	***Solid work***	***Solidedge***	
Type	***Count***	***Count***	***Count***	***Count***	***Solid***	***Surface***
Advanced B-rep Shape rep.	1	1	1	1	1	-
Advanced Face	82	80	76	80	76	-
Axis2 Placement 3d	137	148	138	149	137	197
Cartesian Point	445	450	426	451	423	901
Circle	72	68	60	68	60	120
Closed Shell	1	1	1	1	1	1
Cylindrical Surface	30	34	30	34	30	30
Direction	356	450	422	452	98	686
Edge Curve	228	222	206	222	206	412
Edge Loop	98	96	96	96	96	96
Face Bound	16	16	32	16	16	-
Face surface	-	-	-	-	-	76
Face Outer Bound	82	80	64	80	80	-
Line	156	154	146	154	146	292
Manifold Solid B-rep	1	1	1	1	1	-
Manifold surface shape rep.	-	-	-	-	-	1
Shell based surface model	-	-	-	-	-	1
Oriented Edge	456	444	412	444	412	412
Plane	34	46	46	46	46	46
Vector	156	154	146	154	146	292
Vertex Point	152	148	140	148	140	412

(-): means that feature is absent.

VII. NEUTRAL FORMAT STANDARDS POST-PROCESSOR

The post-processor mission is to recognize the input data (entities) from the pre-processor of the originating system and processed these entities into the format of the receiving CAD system.

A. The Processed Entities

Table 2 shows the final results of both IGES and STEP Post processors for the Model. It should be noticed that the real total number of faces are 76 faces for the model.
IGES post-processors for the model at the receiving systems as shown in table 2 processed the input data from the originating systems as a number of surfaces in most cases.

The number of surfaces processed is between 76 and 80 to 82 for the model. There is a case contains 72 surfaces for the model which contained number of missing surfaces.
STEP post-processors for the model at the receiving systems as shown in table 2 processed the input data from the originating systems as a solid in all of the cases except the import data from Solidedge via surface STEP pre-processor.

For imported model of 80 entities via IGES that happened at the originating system, the circular holes split into two halves each half represent a feature while for imported model 2 of 82 the half circle also split into 2 quarters at the originating system (CATIA) each represent a feature while for imported model 2 of 76 entities each circular hole represent a feature as shown in Fig. 3(a).

TABLE 2. IGES AND STEP POST-PROCESSOR RESULTS FOR THE MODEL.

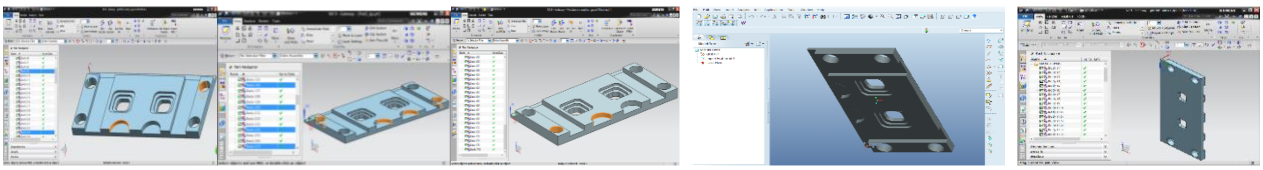
IGES					
From/to	***CATIA***	***Pro/Eng.***	***NX***	***S.W***	***S.E***
CA		82 surface	82 surfaces	1 solid or 82 surfaces	82 surface
P.E.SO	80 trimmed surfaces		80surfaces	1 solid or 80 surfaces	80 surface
P.E.SO	1 solid		1 solid	1 solid	1 solid
NX	76 surfaces	80 surfaces		1 solid or 76 surfaces	76surfaces
S.W.F	80 surfaces	80 surfaces	80 surfaces		80 surfaces
S.W.M	1 solid	80 surfaces	1 solid		1Solid
S.W.B	76 surfaces	72 surfaces	76 surfaces		76 surfaces
S.E.SF	76 surfaces	80 surfaces	76 surfaces	76 surfaces	
S.E.SO	1 solid of 72 surfaces	80 surfaces	1 solid	1 solid	

STEP					
From/to	***CATIA***	***Pro/Eng.***	***NX***	***S.W***	***S.E***
CA		1 manifold solid Brep of 82 faces	1 solid	1 solid of 82 surfaces	1 solid
P.E	1 manifold solid Brep of 80 faces		1 solid	1 solid	1 solid
NX	1 manifold solid Brep of 76faces	1 manifold solid Brep of 80 faces		1 solid of 76 surfaces	1 solid
S.W	1 manifold solid Brep of 80 faces	1 manifold solid Brep of 80 faces	1 solid		1 solid
S.E.SF	76 face surfaces	1 manifold solid Brep of 80 faces	80 surfaces	76 face surfaces	
S.E.SO	1 manifold solid Brep of 76 faces	80 face surfaces	1 solid	1 solid	

B. Data exchange Issues

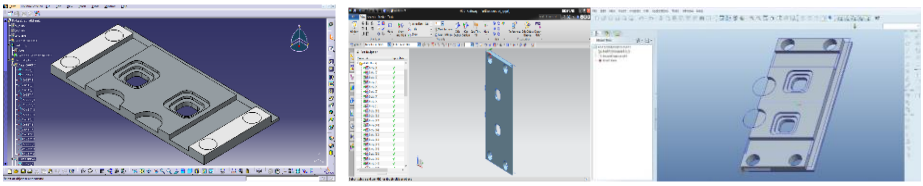
The perfect data exchange results happen when the CAD system import different data formats and deals with them as native data. In all cases of imported data via IGES or STEP, the construction history was lost with all design parameters. Due to this information lost, all of the imported models are dumb solids that is mean cannot be edited. All the systems create a history of the imported features via IGES or STEP except for Pro/Eng., which does not create any history as shown in Fig. 3(b).

The import data of the model from Solidworks and Pro/Eng. to the other systems and vise-versa are represented in different planes from the originating system due to the differences in the axes of the planes as shown in Fig. 3(c). IGES post-processor of CATIA imported the data from IGES solid pre-processor of Solidedge. All the circular holes of model represented as solids as shown in Fig. 4(a). Pro/Eng. and NX IGES post processor imported the design intent and represented as wireframe features from IGES pre-processor of Solidworks than the others as shown in Fig. 4(b).



(a) (b) (c)

Fig. 3. (a) NX import representation, (b) features tree lost for the imported data to Pro/Eng., (c) view plane of NX and Solidworks for the imported data.



(a) (b)

Fig. 4. (a) CATIA representation of model import data via solid IGES from Solidedge, **(**b) NX and Pro/Eng. wireframe import from Solidworks.

C. Imported Total Surface Area

The accuracy status of the surface area at the receiving system compared with the originating system for model is shown in table 3 via IGES and STEP.

It is clear from table 3 that the imported area via STEP was correct in most of the cases except the data imported to CATIA. The surface area of imported data via STEP surface Solidedge failed to import at the Pro/Eng. receiving system.

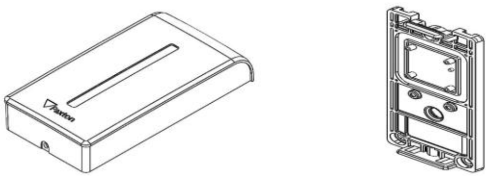
The accuracy percentage of transferring can be calculated for each format by dividing the number of cases with correct surface area by the total cases.

TABLE 3. THE ACCURACY STATUS OF MODEL TOTAL SURFACE AREA-IGES AND STEP.

<table>
<tr><th colspan="6">IGES</th><th colspan="6">STEP</th></tr>
<tr><th>From to</th><th>CATIA</th><th>Pro/Eng.</th><th>NX</th><th>Solidworks</th><th>Solidedge</th><th>From to</th><th>CATIA</th><th>Pro/Eng.</th><th>NX</th><th>Solidworks</th><th>Solidedge</th></tr>
<tr><td>CATIA</td><td></td><td>√</td><td>-</td><td>√</td><td>√</td><td>CATIA</td><td></td><td>√</td><td>√</td><td>√</td><td>√</td></tr>
<tr><td>Pro/Eng. surface</td><td>+</td><td></td><td>+</td><td>√</td><td>√</td><td rowspan="2">Pro/Eng.</td><td rowspan="2">+</td><td rowspan="2"></td><td rowspan="2">√</td><td rowspan="2">√</td><td>√</td></tr>
<tr><td>Pro/Eng. Solid</td><td>√</td><td></td><td>√</td><td>√</td><td>√</td><td>√</td></tr>
<tr><td>NX</td><td>-</td><td>0</td><td></td><td>-</td><td>0</td><td>NX</td><td>-</td><td>-</td><td></td><td>-</td><td>-</td></tr>
<tr><td>Solidworks surface</td><td>-</td><td>√</td><td>+</td><td></td><td>√</td><td rowspan="3">Solidworks</td><td rowspan="3">+</td><td rowspan="3">√</td><td rowspan="3">√</td><td rowspan="3"></td><td rowspan="3">√</td></tr>
<tr><td>Solidworks manifold</td><td>-</td><td>√</td><td>√</td><td></td><td>√</td></tr>
<tr><td>Solidworks bounded</td><td>-</td><td>0</td><td>-</td><td></td><td>√</td></tr>
<tr><td>Solidedge surface</td><td>+</td><td>+</td><td>-</td><td>√</td><td></td><td>Solidedge surface</td><td>+</td><td>0</td><td>-</td><td>√</td><td></td></tr>
<tr><td>Solidedge solid</td><td>-</td><td>+</td><td>√</td><td>√</td><td></td><td>Solidedge solid</td><td>+</td><td>√</td><td>√</td><td>√</td><td></td></tr>
<tr><td colspan="6">√: surface area is the same as at the originating system.
+: surface area is increased from the one at originating system.</td><td colspan="6">- : surface area is decreased from the one at originating system.
0 : failed</td></tr>
</table>

VIII. INDUSTRIAL CASE STUDY

1- Access Control company in the United Kingdom specialized in security access point provided front cover design Fig. 5(a), and the back plate cover design Fig. 5(b), to test.



(a) (b)

Fig. 5. Industrial case studies designs.

2- One of the suggested new designs for the front cover was made by the Lean PPD team/Cranfield University for the company. This design was provided to test and shown in Fig. 6. All the company designs were provided in the IGES form and exported from the Solidworks 2014 CAD system while the suggested design from the lean PPD team was exported from Solidworks 2013 and in the STEP form. These designs imported to all the CAD systems. The numbers of data exchange processes were 5 for each design.

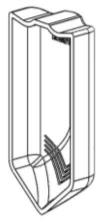


Fig. 6. The suggested design for the front cover.

a. Import Data from Solidworks via IGES

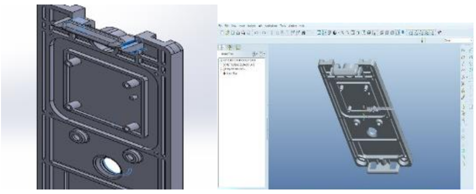
The back plate cover

The input data that provided by the company for the back plate cover is shown in table 4.

TABLE 4. BACK PLATE COVER INPUT DATA (IGES PRE-PROCESSOR FROM SOILDWORKS).

Entity type	Name	Entity count
100	Arc	1609
102	Composite curve	2084
110	Line	3135
120	Surface of revolution	490
124	Transformation matrix	1307
126	Rational B-spline curve	5607
128	Rational B-spline surface	486
142	Curve on a parametric solid	1042
144	Trimmed parametric surface	976

CATIA imported the design as trimmed surfaces (type 144), 975 of 976 surfaces were converted and 1 surface could not be created (degenerated). Solidworks succeeded to import the solid design but indicated that 4 faulty surfaces pierces through the solid as shown in Fig. 7(a), this could be caused by large fillet or chamfer on a thin shell, or by self-intersecting lofting, swept or boundary surface features. Pro/Eng. and Solidedge processed 975 surfaces while NX succeeded to import 976 and all the degeneracy were corrected through the translation. The import representation of all systems is shown in Fig. 7(b).



a. b.

Fig.7. (a) The faulty faces of the imported back plate cover into Solidworks (b) imported back plate cover design.

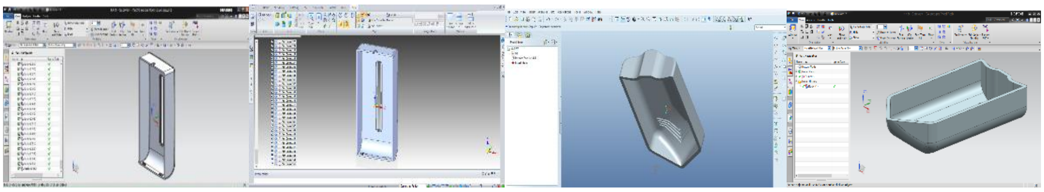
The Front Cover

The input data that provided by the company for the front cover is shown in table 5.

TABLE 5. FRONT COVER INPUT DATA (IGES PRE-PROCESSOR FROM SOLIDWORKS)

Entity Type	*Name*	*Entity Count*
100	Circular Arc	160
102	Composite Curve	548
110	Line	536
120	Surface of Revolution	55
124	Transformation Matrix	148
126	B-Spline Curve	2080
128	B-Spline Surface	201
142	Curve of Surface	274
144	Trimmed Surface	256
314	Colour definition	11

CATIA imported the design as trimmed surfaces (type 144), 251 of 256 surfaces was converted and 5 surface could not be created (degenerated) Solidworks succeeded to import a solid contains 254 of 256 surfaces type (144) and 197 of 201 B-spline surfaces (type 128). The total number of surfaces that could not be created is 6 surfaces. Solidedge processed 251 surfaces while NX succeeded to import 256 and all the degeneracy were corrected through the translation. Pro/Eng. processed only 246 surfaces. The import representation of all the systems is shown in Fig. 8(a).



(a) (b)

Fig. 8. (a) Imported front cover design, (b) imported suggested developed front cover.

b. Import Data from Solidworks via STEP (Industrial Case Study).

The input data that provided by the lean PPD group at Cranfield University for the suggested front cover design is shown in table 6.

TABLE 6. SUGGESTED DESIGN INPUT DATA (STEP PRE-PROCESSOR FROM SOLID WORKS).

Entity Type	*Count*	*Entity Type*	*Count*
Advanced B-rep Shape Representation	1	Face Bound	16
Advanced Face	297	Face Outer Bound	297
Axis2 Placement 3d	318	Line	491
Cartesian Point	4229	Manifold Solid B-rep	1
Circle	86	Spherical surface	12
Closed Shell	1	Toroidal surface	1
Cylindrical Surface	51	Oriented Edge	1634
Direction	1127	Plane	167
Edge Curve	817	Vector	491
Edge Loop	313	Vertex Point	530

All the systems imported the design as a solid feature. The import representation of all the systems is shown in Fig.8(b).

IX. SUGGESTED TOOLS FOR MODIFICATION

A. Feature- Recognition Tool (FRT)

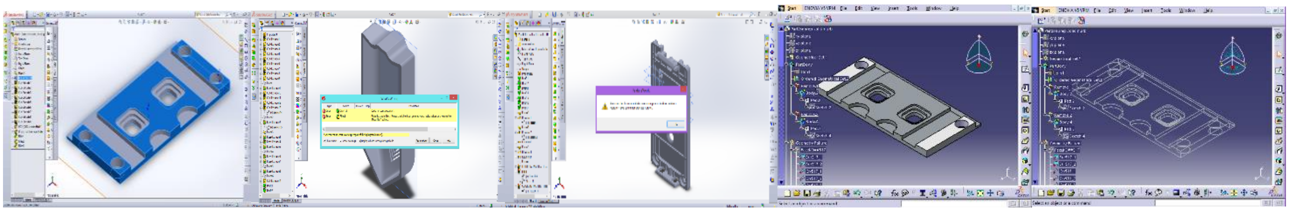
Feature- recognition tool is used to recreate the modeling history from dumb solid via feature recognition. And once the history is created, then work can be done there. The benefit of model feature recognition is that the identified features are updated when the model feature is modified, especially when adding features or faces. The feature recognition implemented in the CAD systems can recognize: Extrudes, Holes, Fillets/Chamfers, Ribs, revolves, volume and drafts. It's clear that the FRT succeed to recognize all the features in the model and easily can modify any feature. The recognized features are 9 cut-extrude circles and fillets Fig. 9 (a).

For the front cover, the FRT failed to extract, only few number of features were recognized. FRT was succeeded to recognize most of the extrude features for the model in Fig. 9(b). For the back plate cover Fig. 9(c) the FRT completely failed to extract the features.

b. Non-Parametric Modeling Method

Major 3D modeling software all have the tools for non-parameter modeling, if any model was imported from IGES or STEP, all parameters have been removed, then the non-parameter base method is needed to edit this hole. One simple way is measuring the diameter of the current hole first, and using the offset face tool to offset the hole surface to the diameter which is required. The other way is just deleting current surface of the hole and creating a new one. the same example above, one other method is that using the extrude method to generate a solid block to cover the hole volume, it has the same result as delete the hole surface, and dig a new hole with parameter (the required diameter) as applied to the model import data to CATIA from Solidworks via solid IGES Fig. 9(d). All systems succeeded to modify any kind of modification from changing the diameters of holes to adding/remove faces by using non parametric method.

The non-parametric method worked well with the solid body while for surface body there are some limitations.



(a) (b) (c)

Fig. 9. (a) FRT for the tested model, (b) FRT for the industrial case studies, (c) adding hole feature using non-parametric method.

X. CONCLUSIONS

The analysis of the CAD data exchange was presented and found that STEP is more effective than IGES with solid translation while IGES is more effective with surface translation. Some of the cases of the imported data via IGES suffered from missing surfaces. STEP succeeded to transfer 58% correctly of the imported total surface area cases while 50% of the cases via IGES. The file size of STEP format is much less than IGES for all the pre-processors of the systems for the same samples. For the industrial case studies, the best import data was to NX while the worst import with the most missing surfaces happened with Pro/Eng. All imported data lost the construction history of the designs and the systems deal with them as dumb solids. The non-parametric modeling method is more effective than feature recognition tool to deal with dumb solid for modification.

REFERENCES

[1] G. Tassey, S. Brunnermeier and S. Martin,"Interoperability cost analysis of the US automotive supply chain", Research Triangle Institute Final Report. RTI Project, 1999.

[2] J. Liu, W. Ming, Z. Jun, W. Kun and Z. m. Chen, "A method for data exchange between feature-based CAD models", Intelligent Control and Automation (WCICA), 2010 8th World Congress on, IEEE, 2010.

[3] H. Daniyal, "Study on interoperability problems among CAD/CAM systems in automotive Industry", M.Sc.Thesis, Kol Universiti Teknologi Tun Hussein Onn, Malaysia, 2004.

[4] M. Bhandarkar, B. Downie, M. Hardwick and R. Nagi, "Migrating from IGES to STEP: one to one translation of IGES drawing to STEP drafting data", Computers in Industry, 2000, pp.261-277.

[5] A. Kamrani and E. Nasr, "Rapid prototyping: theory and practice", Springer, 2006.