



# The CAD Cross-reference Tool for Pro/E and Solidworks Users

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## NX3 Unigraphics

### Datum Tools

#### How to :

Create Datum plane.	Insert > Datum/ point > Datum plane > OK
offset	Insert > Datum/ point > Datum plane >Constraints Cascade Menu > offset > OK
angular offset	Insert > Datum/ point > Datum plane >Constraints Cascade Menu > angle > OK
Create Datum Axis.	Insert > Datum/ point > Datum axis > OK
Create Datum point on:	
Curve	Insert > Datum/ point > Point Set > Points on Curve > OK
Edge	Insert > Datum/ point > Point Set > Points on Curve > OK
Datum Axis	N/A
Curve's Intersection	Insert > Datum/ point > Point Set > Points on Curve > OK
At Center	Insert > Datum/ point > Point > Select Arc/ Circle > OK
On/ Offset from a surface	Insert > Datum/ point > Point Set > Points on face > OK
Offset from other points	Insert > Datum/ point > Point Set > Points on face > OK
Intersecting entities	Insert > Datum/ point > Point > Points on intersection > OK
Create Sketched Datum Curve	Insert > Curve > Pick type of line to sketch > OK
Create Coordinate System	
Locate	Insert > Datum CSYS > Pick type > OK
Offset	Insert > Datum CSYS > Pick Offset type in CSYS Constructor > OK
Orient	Insert > Datum CSYS > Pick Oriented type in CSYS Constructor > OK
From File	N/A

### Feature Creation Tools

#### How to :

Create an Extrusion	
Solid	Insert > Design Feature > Extrude > Select sketched geometry > OK
Thick	Insert > Design Feature > Extrude > Select sketched geometry > Offset to thicken > OK
Cut	Insert > Design Feature > Extrude > Select sketched geometry > Boolean Subtract > OK
Surface	Insert > Design Feature > Extrude > Select sketched open geometry > Offset to thicken > OK
Surface Trim	Insert > Design Feature > Extrude > Select sketched geometry > Boolean Subtract > OK
Revolved Solid	Insert > Design Feature > Revolve > follow cues > OK
Thick Revolved Feature	Insert > Design Feature > Revolve > follow cues > OK
Revolved Cut	Insert > Design Feature > Revolve > Boolean Subtract, follow cues > OK
Revolved Surface	Insert > Design Feature > Revolve > Offset, follow cues > OK
Revolved Surface Trim	Insert > Design Feature > Revolve > Boolean Subtract, follow cues > OK
Variable Section Sweep	Insert > Sweep > Swept > follow cues > OK
Create a Blend	
Basic	Insert > Detail feature > Edge Blend > follow cues > OK
Import	N/A

### Edit Feature Tools

#### How to :

Copy a Feature	
Surface	Edit > Copy feature > select surface > OK
Without Holes and Patches	Edit > Copy feature > select surface > OK
Bounded by a Closed Curv	Edit > Copy feature > select surface > OK
Curves	Edit > Copy feature > select curve > OK
Edges	Edit > Copy feature > select edge > OK
Mirror	
Part Geometry	Insert > Associative Copy > Instance > Mirror Body > follow cues > OK
Selectable Features	Insert > Associative Copy > Instance > Mirror feature > follow cues > OK

### Miscellaneous Tools

#### How to :

Move Entities	
Surface	Edit > Feature > Move > Select surface > move > OK
Curves	Edit > Feature > Move > Select curve > move > OK
Axis	Edit > Feature > Move > Select axis > move > OK

## NX3 Unigraphics

<b>Rotate Entities</b>	
Surface	Edit > Feature > Move > Select surface > rotate between two axes > OK
Curves	Edit > Feature > Move > Select curve > rotate between two axes > OK
<b>Merge Entities</b>	
Selected Features	Insert > Combine Bodies > Unite > OK
<b>Trim Entities</b>	
Curves	Insert > Combine Bodies > Subtract > OK
Surfaces	Insert > Combine Bodies > Subtract > OK
<b>Pattern Features</b>	
Dimension	Insert > Associative Copy > Instance > Mirror feature > follow cues > OK
Table	Insert > Associative Copy > Instance > Mirror feature > follow cues > OK
Reference	Insert > Associative Copy > Instance > Mirror feature > follow cues > OK
<b>Projections</b>	
Datum Curves	
By sketch	Insert > Curve from Curves > Project > follow cues > OK
By selection	Insert > Curve from Curves > Project > follow cues > OK
<b>Create a Wrap</b>	
Datum Curves	Insert > Offset/scale > Wrap geometry > OK
<b>Create Extensions</b>	
Surface	Insert > Trim > Trim and Extend > choose surface > OK
Tangent Surfaces	Insert > Trim > Trim and Extend > choose surface > OK
To a Plane	Insert > Trim > Trim and Extend > choose surface > OK
<b>Create Intersections from</b>	
Sketches	Insert > Curve from Bodies > Intersect > select sketches > OK
Surfaces	Insert > Curve from Bodies > Intersect > select surface > OK
Create Fills	Insert > Combine Bodies > Patch OR Insert > Surface > Transition > OK for Non planar surfaces
Redefine a Fills	Select Transition or Patch in Feature tree > right click > Edit > follow cues > OK
<b>Create Offset</b>	
Surface	
Standard	Insert > Offset/ scale > offset > select surfaces > OK
Expanded	Insert > Offset/ scale > offset > select surfaces > OK
Constraints	Insert > Offset/ scale > offset > select surfaces > OK
Drafts	Insert > Offset/ scale > offset > select surfaces > OK
Replacing	Insert > Offset/ scale > offset > select surfaces > OK
Curves	
Normal	Insert > Offset/ scale > offset > select curve > OK
Along	Insert > Offset/ scale > offset > select curve > OK
Fan Curve	Insert > Offset/ scale > offset > select curve > OK
Boundary	Insert > Offset/ scale > offset > select curve > OK
Variable Boundary	Insert > Offset/ scale > offset > select curve > OK
Create a Thicken	Insert > Offset/ scale > Thicken sheet > select surface > OK
<b>Create a Solid on surfaces</b>	
Protrusion	Insert > Offset/ scale > Sheets to a solid assistant > select surface> Boolean Create > OK
Cut	Insert > Offset/ scale > Sheets to a solid assistant > select surface> Boolean Subtract > OK
Patches	Insert > Offset/ scale > Sheets to a solid assistant > select surface> Boolean Unite > OK
<b>Create Holes</b>	
Sketched	Insert > Design Feature > Extrude subtract > Sketch hole, follow cues > OK
Simple	Insert > Design Feature > Hole > Simple, follow cues > OK
Counterbore	Insert > Design Feature > Hole > Counterbore, follow cues > OK
Countersunk	Insert > Design Feature > Hole > Countersunk, follow cues > OK
<b>Create Shells</b>	
Shells	Insert > Offset/ scale > Hollow > select surface/s to pierce > OK
<b>Create Ribs</b>	
Ribs	Use freeform features to create.
<b>Create Drafts</b>	
Standard	Insert > Detail feature > Taper select type> OK
Variable	Insert > Detail feature > Taper select type> OK
Split	Insert > Detail feature > Taper select type> OK
Split with 2 Hinge Planes	Insert > Detail feature > Taper select type> OK
<b>Create Rounds</b>	
Basic	Insert > Detail feature > Edge Blend select type > OK
<b>Create Chamfers</b>	
Basic	Insert > detail feature > chamfer > OK



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## Pro-Engineer Wildfire

### Datum Tools

#### How to :

Create Datum plane.	Insert > Model Datum > Plane> choose constraints> OK
offset	Insert > Model Datum > Plane> select existing Plane/ planar surface> OK
angular offset	Insert > Model Datum > Plane> select existing Axis/ Edge> select Through> Offset Rotation > OK
Create Datum Axis.	Insert > Model Datum > Axis> select two references (planes, surfaces, points, curves)> OK
Create Datum point on:	
Curve	Select an entity (Curve) > select points Icon>Placement > OK
Edge	Select an entity (edge) > select points Icon>Placement > OK
Datum Axis	Select an entity (Datum axis)> select point Icon>Placement > OK
Curve's Intersection	Select an entity (2 curves)> select point Icon>Placement > OK
At Center	Select an entity (arc, circular geometry)> select point Icon>Placement > OK
On/ Offset from a surface	Select an entity (surface)> select point Icon>Placement > in the offset dimension box type value> OK
Offset from other points	Select an entity (point)> select point Icon>Placement > in the offset dimension box type value> OK
Intersecting entities	Select 2 intersecting entities> select point Icon>Placement > in the offset dimension box type value> OK
Create Sketched Datum Curve	Insert > Model Datum > Sketched Curve
Create Coordinate System	
Locate	Insert > Model Datum > Coordinate System
Offset	Insert > Model Datum > Coordinate System > click offset Tab type x,y,z distances > OK
Orient	Insert > Model Datum > Coordinate System > in orient Tab > OK
From File	Insert > Model Datum > Coordinate System > Select from file in menu > OK

### Feature Creation Tools

#### How to :

Create an Extrusion	
Solid	Insert > Extrude > follow cues > OK
Thick	Insert > Extrude > follow cues > click on the thicken button> OK
Cut	Insert > Extrude > follow cues > click on the remove material button> OK
Surface	Insert > Extrude > follow cues > click on the surface button> OK
Surface Trim	Insert > Extrude > follow cues > click on the surface button> click remove material button> OK
Revolved Solid	Insert > Revolve > follow cues > OK
Thick Revolved Feature	Insert > Revolve > follow cues > click on the thicken button> OK
Revolved Cut	Insert > Revolve > follow cues > click on the remove material button> OK
Revolved Surface	Insert > Revolve > follow cues > click on the surface button> OK
Revolved Surface Trim	Insert > Revolve > follow cues > click on the surface button> click remove material button> OK
Variable Section Sweep	Insert > Variable Section Sweep > click surface/ solid button > follows cues > OK
Create a Blend	
Basic	Insert> Blend > blend opts > Done
Import	Insert> Blend > blend opts > from file > Done

### Edit Feature Tools

#### How to :

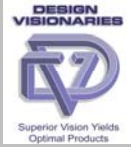
Copy a Feature	
Surface	Click Edit > Copy > OK
Without Holes and Patches	Click Edit > Copy > right click Exclude Surfaces and fill holes > OK
Bounded by a Closed Curve	Click Edit > Copy > right click Copy Inside Boundary > OK
Curves	Click Edit > Copy > right click Approximate/ Exact > OK
Edges	Click Edit > Copy > right click Approximate/ Exact > OK
Mirror	
Part Geometry	Select part name from the Model Tree > Edit > Mirror > select mirror plane > OK
Selectable Features	Select features > Edit > Mirror > select mirror plane > OK

#### How to :

Move Entities	
Surface	Select surface > Edit > Move > translate, follow cues > OK
Curves	Select curve > Edit > Move > translate, follow cues > OK
Axis	Select axis > Edit > Move > translate, follow cues > OK

## Pro-Engineer Wildfire

Rotate Entities	
Surface	Select surface > Edit > Move > Rotate, follow cues > OK
Curves	Select curve > Edit > Move > Rotate, follow cues > OK
Merge Entities	
Selected Features	Select 2 surfaces > Edit > Merge > Select method to merge, follow cues > OK
Trim Entities	
Curves	Select Curve > Edit > Trim > Select direction, follow cues > OK
Surfaces	Select Surfaces > Edit > Trim > Select Options Thin Trim, follow cues > OK
Pattern Features	
Dimension	Select Feature to pattern > Edit > Pattern > Select dimension to pattern to, follow cues > OK
Table	Select Feature to pattern > Edit > Pattern > Select table as method , follow cues > OK
Reference	Select Feature to pattern > Edit > Pattern > Select reference as method , follow cues > OK
Projections	
Datum Curves	
By sketch	Edit > Project > References tab > Select a sketch > follow cues > OK
By selection	Edit > Project > References tab > Select chains > follow cues > OK
Create a Wrap	
Datum Curves	Edit > Wrap > click Destination > follow cues > OK
Create Extensions	
Surface	Select edges of Surfaces > Edit > Extend > Select Options Same, follow cues > OK
Tangent Surfaces	Select edges of Surfaces > Edit > Extend > Select Options Tangent, follow cues > OK
To a Plane	Select edges of Surfaces > Edit > Extend > Select Options To Plane, follow cues > OK
Create Intersections from	
Sketches	Edit > Definition > Collect sketch, follow cues > OK
Surfaces	Edit > Definition > Collect surface, follow cues > OK
Create Fills	
Redefine a Fills	Select fill feature in Tree > right click > Edit Definition > click continue, follow cues > OK
Create Offset	
Surface	
Standard	Edit > Offset > enter value for offset > select standard for type > OK
Expanded	Edit > Offset > enter value for offset > select expand for type > OK
Constraints	Edit > Offset > enter value for offset > select expand for type > sketched Region > OK
Drafts	Edit > Offset > enter value for offset > select with drafts for type > select sketch, follow cues > OK
Replacing	Edit > Offset > enter value for offset > select replace for type > select sketch, follow cues > OK
Curves	
Normal	Edit > Offset > enter value for offset > Reference tab > select Normal to surface, follow cues > OK
Along	Edit > Offset > enter value for offset > Reference tab > select Along to surface, follow cues > OK
Fan Curve	Edit > Offset > Reference tab > select Parallel to plane, follow cues > OK
Boundary	Edit > Offset > enter value for offset > Reference tab > , follow cues > OK
Variable Boundary	Edit > Offset > enter value for offset > Reference tab > , follow cues > OK
Create a Thicken	
Create a Solid on surfaces	
Protrusion	Edit > Solidify > select surface > Add material, follow cues > OK
Cut	Edit > Solidify > select surface > Remove material, follow cues > OK
Patches	Edit > Solidify > select surface > Remove/ Add material, follow cues > OK
Create Holes	
Sketched	Insert > Hole > Sketched > follow cues > OK
Simple	Insert > Hole > click Simple Icon > follow cues > OK
Counterbore	Insert > Hole > click Counterbore Icon > follow cues > OK
Countersunk	Insert > Hole > click Countersunk Icon > follow cues > OK
Create Shells	
Shells	Insert > Shell > follow cues > OK
Create Ribs	
Ribs	Insert > Rib > follow cues > OK
Create Drafts	
Standard	Insert > Draft > Select hinges > follow cues > OK
Variable	Insert > Draft > Select hinges > follow cues > OK
Split	Insert > Draft > Select hinges > select split, follow cues > OK
Split with 2 Hinge Planes	Insert > Draft > Select hinges > select split by split object, follow cues > OK
Create Rounds	
Basic	Insert > Round > enter round dimension > follow cues > OK
Create Chamfers	
Basic	Insert > Chamfer > Edge Chamfer, follow cues > OK



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## Solidworks 2004

### Datum Tools

#### How to :

Create Datum plane.	Insert > Reference Geometry > Plane.
offset	Insert > Reference Geometry > Plane > Offset
angular offset	Insert > Reference Geometry > Plane > At Angle
Create Datum Axis.	Insert > Reference Geometry > Axis
Create Datum point on:	
Curve	Insert > Reference Geometry > Point > select curve, follow cues > OK
Edge	Insert > Reference Geometry > Point > select edge, follow cues > OK
Datum Axis	Insert > Reference Geometry > Point > select axis, follow cues > OK
Curve's Intersection	Insert > Reference Geometry > Point > select intersection, follow cues > OK
At Center	Insert > Reference Geometry > Point > select center of face, follow cues > OK
On/ Offset from a surface	Insert > Reference Geometry > Point > select projection, follow cues > OK
Offset from other points	Insert > Reference Geometry > Point > select along curve, follow cues for distance > OK
Intersecting entities	Insert > Reference Geometry > Point > select intersection, follow cues > OK
Create Sketched Datum Curve	Insert > Curve > make it a datum > OK
Create Coordinate System	
Locate	Insert > Reference Geometry > Coordinate System > OK
Offset	Insert > Reference Geometry > Coordinate System > type in x,y,z values > OK
Orient	Insert > Reference Geometry > Coordinate System > reverse axis direction > OK
From File	Insert > Reference Geometry > Coordinate System > OK

### Feature Creation Tools

#### How to :

Create an Extrusion	
Solid	Insert > Boss/Base > Extrude > OK
Thick	Insert > Boss/Base > Extrude > select Thin feature > OK
Cut	Insert > Cut > Extrude > input end conditions > OK
Surface	Insert > Boss/Base > Extrude > select thin > input end conditions > OK
Surface Trim	Insert > Cut > With Surface > Follow cues > OK
Revolved Solid	Insert > Boss/Base > Revolved > select axis of rotation > input end conditions > OK
Thick Revolved Feature	Insert > Boss/Base > Revolved > select thin feature > select axis of rotation > input end conditions > OK
Revolved Cut	Insert > Cut > Revolved > follow cues > OK
Revolved Surface	Insert > Surface > Revolve > select axis > select add material > follow cues > OK
Revolved Surface Trim	Insert > Surface > Revolve > select axis > select remove material > follow cues > OK
Variable Section Sweep	Insert > Boss/Base > Sweep > follow cues > OK
Create a Blend	
Basic	Insert > Features > Fillet/Round > select type > enter radius > follow cues > OK
Import	

### Edit Feature Tools

#### How to :

Copy a Feature	
Surface	Edit > Copy > Select surface > click where you want to paste surface > paste > OK
Without Holes and Patches	N/A
Bounded by a Closed Curv	N/A
Curves	Edit > Copy > Select curve > click where you want to paste curve > paste > OK
Edges	Edit > Copy > Select edge > click where you want edge > paste > OK
Mirror	
Part Geometry	Insert > Pattern/Mirror > Mirror > select plane > select parts to be mirror > OK
Selectable Features	Insert > Pattern/Mirror > Mirror > select plane > select features to be mirror > OK

#### How to :

Move Entities	
Surface	Insert > Surface > Move/ Copy > follow cues > OK
Curves	Insert > Features > Move/ Copy > follow cues > OK
Axis	Click on Axis > Right click > Edit feature > follow cues > OK

## Solidworks 2004

<b>Rotate Entities</b>	
Surface	Insert > Move/ Copy Bodies > Select surface > rotate > OK
Curves	Insert > Move/ Copy Bodies > Select curve > rotate > OK
<b>Merge Entities</b>	
Selected Features	Insert > Features > Combine > select entities > OK
<b>Trim Entities</b>	
Curves	Insert > Features > Split > select curve > OK
Surfaces	Insert > Features > Split > select surface > OK
<b>Pattern Features</b>	
Dimension	Insert > Pattern/ Mirror > linear > OK
Table	Insert > Pattern/ Mirror > load table pattern coordinates > OK
Reference	Insert > Pattern/Mirror > Sketch Driven Pattern > Use sketch as reference > OK
<b>Projections</b>	
Datum Curves	
By sketch	Insert > Curve > Projected > follow cues > OK
By selection	Insert > Curve > Projected > select curves follow cues > OK
<b>Create a Wrap</b>	
Datum Curves	Insert > Feature > Wrap > follow cues > OK
<b>Create Extensions</b>	
Surface	Insert > Surface > Extend > follow cues > OK
Tangent Surfaces	Insert > Surface > Extend > follow cues > OK
To a Plane	Insert > Surface > Extend Up to a surface or plane > follow cues > OK
<b>Create Intersections from</b>	
Sketches	In sketcher Tools > Sketch Tools > Intersection Curve > OK
Surfaces	In sketcher Tools > Sketch Tools > Intersection Curve > OK
<b>Create Fills</b>	
Redefine a Fills	In Part tree > right click and select Edit feature > OK
<b>Create Offset</b>	
Surface	
Standard	Insert > Surface > Offset > set distance > OK
Expanded	Insert > Surface > Offset > set distance > OK
Constraints	Insert > Surface > Offset > set distance > OK
Drafts	Insert > Surface > Offset > set distance > OK
Replacing	Insert > Surface > Offset > set distance > OK
Curves	
Normal	In sketcher click on Offset Entities tool > Pick edges and distance > OK
Along	In sketcher click on Offset Entities tool > Pick edges and distance > OK
Fan Curve	In sketcher click on Offset Entities tool > Pick edges and distance > OK
Boundary	In sketcher click on Offset Entities tool > Pick edges and distance > OK
Variable Boundary	In sketcher click on Offset Entities tool > Pick edges and distance > OK
<b>Create a Thicken</b>	
	Insert > Base (or Boss) > Thicken.
<b>Create a Solid on surfaces</b>	
Protrusion	Insert > Surface > Fill pick solid > OK
Cut	Insert > Surface > Fill pick remove material > OK
Patches	Insert > Surface > Fill pick remove/ Add material > OK
<b>Create Holes</b>	
Sketched	Insert > Sketch > draw hole to extrude > set end conditions > remove material > OK
Simple	Insert > Features > Hole > Simple > OK
Counterbore	Insert > Features > Hole > Counterbore > OK.
Countersunk	Insert > Features > Hole > countersunk > OK.
<b>Create Shells</b>	
Shells	Insert > Features > Shell > OK
<b>Create Ribs</b>	
Ribs	Insert > Features > Rib > OK
<b>Create Drafts</b>	
Standard	Insert > Features > Draft > follow cues neutral > OK
Variable	Insert > Features > Draft > follow cues > OK
Split	Insert > Features > Draft > follow cues > OK
Split with 2 Hinge Planes	Insert > Features > Draft > follow cues > OK
<b>Create Rounds</b>	
Basic	Insert > Features > Fillet/Round > OK
<b>Create Chamfers</b>	
Basic	Insert > Features > Chamfer > OK