

AutoCad Version 14+ thru Current

Once the part or assembly is created in 3D and is in positive space:

1. Type (at the command prompt): **FACETRES**
2. Set facetres Between 1-10 (10 being best)
3. Type (at the command prompt): **STLOUT**
4. Select the objects to be exported
5. Select: **Y for Binary**

CATIA

STL parts created as solids should be fine. If the part was created as volumes from surfaces may create problems. We recommend using the CATIA toolbox to analyze all STL's once they have been generated.

1. Select **STL Command**
2. Maximum Sag = **.0003" or .0125 mm**
3. **Select part(s)** to be converted and Click YES
4. Select **Export**
5. Type **filename and output the STL.**



INVENTOR

Once the part or assembly is created in 3D

1. Select: **Save Copy As**
2. Select: **STL**
3. Select Options Menu: **Set To High**
4. Enter **Filename**
5. **Save**

IRONCAD

Parts must be created in 3D

1. **Right Click** on the Part to be saved
2. Under Part Properties Select **Rendering**
3. Set Facet Surface Smoothing to: **150**
4. Select File: **Export**
5. Select: **STL**



MECHANICAL DESKTOP

1. **AMSTLOUT @** Command Line
2. Angular Tolerance = **0**
3. Aspect Ratio = **0**
4. Surface Tolerance = **.0005" or .0125 mm**
5. Vertex Spacing = **0**

PRO/ENGINEER

1. **FILE / EXPORT / MODEL**
2. Choose **STL**
3. Chord Height: **.0005" or .0125 mm**
4. Angle Control: **.5**
5. Click: **APPLY**

SDRC I-DEAS

1. **FILE / EXPORT / RAPID PROTOTYPE / OK**
2. Select the **Part**
3. Select: **PROTOTYPE DEVICE / SLA500.dat / OK**
4. Absolute Facet Deviation: **.0004**
5. Select Binary / **OK**

Rhino

Parts must be created in 3D

1. **FILE / SAVE AS**
2. Select File Type: **STL**
3. Enter a **File Name** and **Save**
4. Select **Binary File**

SolidDESIGNER

1. **FILE / SAVE / STL**
2. Set to: **Binary**
3. Select **Part**
4. Set Max Deviation Distance to: **.001 mm**
5. Select: **OK**

SOLIDEDGE

1. **FILE / SAVE AS**
2. Set Save as Type: **STL** then select **Options**
3. Set: **Conversion Tolerance: .0005" or .0125mm**
4. Set Surface Plane to: **45.00** (degrees)
5. **Save**

SOLIDWORKS

1. **FILE / SAVE AS**
2. **SAVE AS TYPE / Select: STL**
3. Select: **Options**
4. Deviation Tolerance: **.0004"**
5. Angle Tolerance: **7.75 deg**
6. **SAVE**

UNIGRAPHICS

1. **FILE / EXPORT / Rapid Prototyping**
2. Choose **Binary**
3. Triangle Tolerance: **.0005" or .0125 mm**
4. Adjacency Tolerance: **0.12**
5. Click: **APPLY**
6. Set Auto Normal Gen to: **ON**
7. Set Normal Display to: **OFF**
8. Set Triangle Display to: **ON**