

Shape Interrogation for CAD/CAM

offsetc.c

Find an offset curve for an input open NURBS curve by evaluating an offset curve at a specified (nsegs) number of points on the input curve.

Do:

```
prompt> make offsetc
```

```
prompt> offsetc -i input_curve_file -n number_of_segments_per_knot_span -d  
offset_distance -o output_file
```

Output: Resulting offset curve in .VECT format

Example:

```
prompt> offsetc -i c.CURV -n 50 -d 1.0 -o offsetc.VECT
```

Note: For the file format of the input curve (c.CURV) and the output (offsetc.VECT), see [../README.pdf](http://www.cadcam.org/README.pdf)

offsets.c

Find an offset surface for an input open NURBS surface by evaluating an offset surface at a specified (nsegu x nsegv) number of points on the input surface.

Do:

```
prompt> make offsets
```

```
prompt> offsets -i input_surface_file -m number_of_segments_per_u-knot_span -n  
number_of_segments_per_v-knot_span -d offset_distance -o output_file
```

Output: Resulting offset surface in .VECT format

Example:

```
prompt> offsets -i s.SURF -m 20 -n 20 -d 0.3 -o offsets.VECT
```

Note: For the file format of the input surface (s.SURF) and the output (offsets.VECT), see [../README.pdf](http://www.cadcam.org/README.pdf)