

New Version 10.2

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VVD is now in compliance with ASME VIII:2007 A09, EN13445:2009 Issue 1, PD5500:2009 Amend.No.1, EN13480-3:2002/A3:2009..

Version 10.2 – Updates to latest code amendment

Version 10.2 includes a number of improvements and additions. This version is also in compliance with:

- ASME VIII:2007 edition, Addenda 2009

- EN13445:2009 Issue 1

- PD5500:2009 Amendment No. 1

- EN13480-3:2002/A3:2009

The material libraries have been updated to ASME II Part D:2007 A09, and all EN materials have been updated to latest revisions of standards.

Version 10.2 – General

- Added EN13445 section 19, creep design. Included two new test groups for creep design (1c

and 3c), included hydro test requirements for creep design, included table of NDT requirements for creep design. Creep properties from the following EN material standards have been added: EN10028-2, EN10028-7, EN10273, EN10216-2, EN10222-2 and EN10213.

- Added MDMT requirements in ASME VIII Div.1 Section UHA / Appendix JJ.

- Added a new option to attach skirt support to hemispherical heads.

- Added a new option for design of tailing lugs welded into the base plate/upper ring of the skirt support.

New Application: 3D CADConverter

The new 3D CADConverter will use the 3D drawing generated by the VVD software and produce a STEP file that can be used to import the 3D drawings into end-users 3D application. The 3D drawing production process will then be as follows:

VVD Software -> 3D CADConverter -> STEP FILE -> 3D Application

This new 3D CADConverter should allow a rapid drawing production resulting in significant time savings as all elements (or a selection of elements) designed in the VVD software will automatically be exported to the end user 3D application.

The new 3D CADConverter application can be downloaded from our website.

To generate a STEP file, please follow the steps below;

Open/retrieve the model that shall be exported from VVD

Go to 'Action' or '3D Drawing' and select 'Export 3D Drawing to STEP file format.

The generated STEP file will be given the same filename and stored in the same directory as the

model you have opened, the new file will be given STEP as a file extension.