INHERENT POSSIBILITIES AND LIMITATIONS OF FINITE ELEMENT MODELLING OF DEFECTIVE GIRTH WELDS

S. Hertelé[1], W. De Waele[2], R. Denys[2], M. Abdel Wahab[2] and M. Verstraete[2]

[1] FWO Aspirant, Ghent University, Laboratory Soete, Belgium
[2] Ghent University, Laboratory Soete, Belgium

Abstract Welds unavoidably show defects, which can negatively affect the integrity of the entire structure and, worst case, result in a failure. Defects of a considerable size should therefore be detected, assessed and, if necessary, repaired. The assessment of a defect requires a procedure which allows a conservative estimation of the acceptability of the defect. To develop such procedure, both experimental and numerical research is performed. This paper describes the inherent possibilities and limitations of numerical research through finite element modelling, as compared to experimental research. Summarizing all arguments, it becomes clear that numerical research is a highly powerful tool, but a thorough experimental validation is of paramount importance. Moreover, some specific weld-related problems are highly difficult to address, namely the presence of stable crack growth and material heterogeneity. More research is needed to achieve a description of these phenomena, under a set of conservative assumptions.

Keywords

Finite element analysis, weld, defect

1 INTRODUCTION

Welding is a joining technique which, if appropriately performed, has a lot of advantages as opposed to other joining techniques. Noteworthy are the achievement of a light, stiff, fully sealed connection and the possibility for automation. Often, welding is the only economically and practically acceptable joining technique, e.g. in the case of large-diameter, high pressure transport pipelines.

Despite the advantages related to welding, a weld is often the most critical area of the structure, because of two reasons (Figure 1). Firstly, a weldment consists of a continuous transition between different microstructures, some of which show detrimental properties (reduced fracture toughness, reduced strength). Secondly, welds that are made in the field unavoidably contain defects, such as porosities, slag inclusions, lack of fusion.

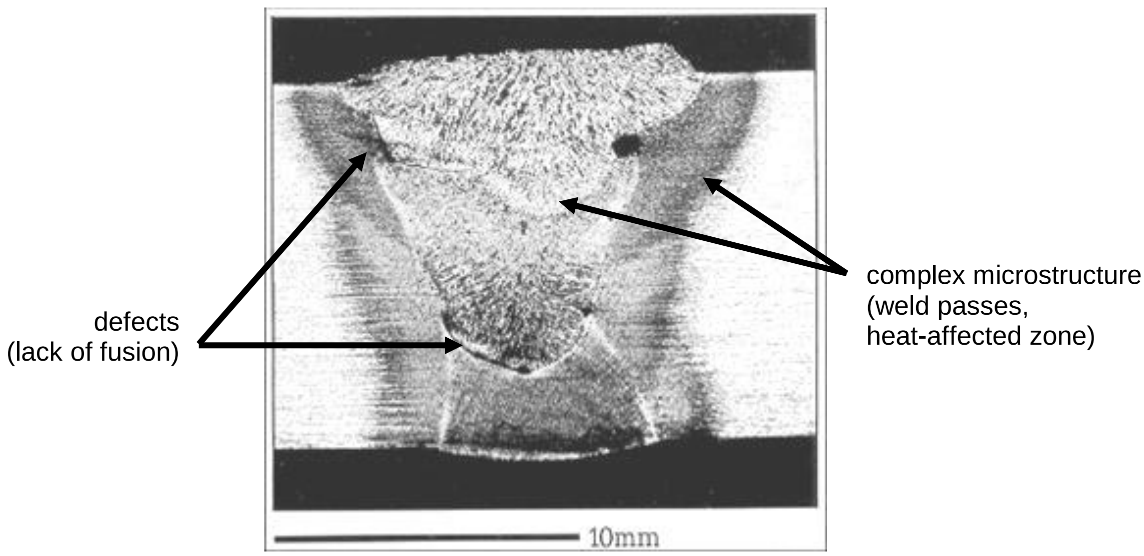

Figure 1: A weldment consists of complex microstructures and defects [1].

 Copyright © 2010 by Labo Soete

Weld defects give rise to stress concentrations, which can lead to crack initiation, followed by stable or unstable crack propagation. In the worst case, one single defect can result in a failure of the global structure, possibly causing casualties and ecological damage. A well-known example that illustrates the possible consequences of welding defects is the collapse of the Alexander Kielland oil platform (123 casualties) [1].

Since weld defects can have disastrous consequences, each structural weld should be non-destructively inspected and, if necessary, repaired. However, a 100% repair rate would be way too expensive. Therefore, defects should be assessed for their acceptability and, if allowable, not be act upon.

The assessment of defects requires an analytical procedure which allows to conservatively estimate their possible influence on the structural integrity. To develop such procedures, which are often application-specific, a lot of experiments have been and are still being performed.

Besides the analysis of experiments, however, there is a second possible approach: finite element analysis. For the specific case of pipeline welds, this approach has been widely applied for two general purposes. First, finite element analyses have been performed to investigate the qualitative effect of a limited set of parameters (descriptive). For instance, as regards the operational conditions, studies have been performed about the loading mode (bending or tensile) [2] and the presence of internal pressure [2,3,4]. Concerning the geometry, the structure's and the defect's size and geometry have been investigated [2-7]. Of particular importance in this category is the effect of weld misalignment [2;4;5]. Another important particular topic of investigation is the material behaviour, including strain hardening [3;6;7], weld strength mismatch [2;6-9], heat affected zone softening [4;9;10], resistance to crack growth [2;6;7;11-14], anisotropy [5] and toughness [5]. Second, finite element analyses have been used to describe an experiment as good as possible, including all important aforementioned aspects related to defective welds (predictive). This category is best represented by ongoing numerical research from ExxonMobil [15-19], and also includes some less extensive finalised projects [20,21].

Since the numerical alternative is fundamentally different to the experimental approach, it has some specific additional advantages, but also some drawbacks. This paper is concerned with the discussion of the positive and negative aspects of finite element analysis for the purpose of assessing pipeline girth weld defects, and how to achieve an optimal research programme. The article is structured as follows. Firstly, the possibilities of finite element modelling are discussed. Then, the limitations are investigated, along with a discussion on how to deal with them. Finally, conclusions are drawn.

2 POSSIBILITIES OF FINITE ELEMENT MODELLING OF PIPELINE GIRTH WELDS

As compared to experiments, finite element simulations provide advantages at three levels: analysis, flexibility and control. Each advantage is separately discussed in the sections below.

2.1 Degree of analysis

Unavoidably, the results of an experiment are confined to the extent and possibilities of the applied instrumentation. Mostly, this instrumentation gives information about a number of deliberately chosen, representative quantities, but not about the entire picture. In some specific cases, this can even give rise to unknown measurement errors [22]. Numerical simulations, on the other hand, can provide the complete space-time response of the investigated problem (Figure 2).

Apart from some basic quantities (stresses, strains, …), other related results can also be obtained. Given as an example is the J integral, which expresses the urge of a crack to initiate and propagate ('crack driving force'), and which can therefore be used to predict crack growth. This quantity is calculated as an integral over a random path around the crack tip, as first defined by Rice [23]. An analytical expression for J is known for only a limited number of standard experimental setups. In other setups, numerical solutions need to be addressed to obtain J.

 Copyright © 2010 by Labo Soete

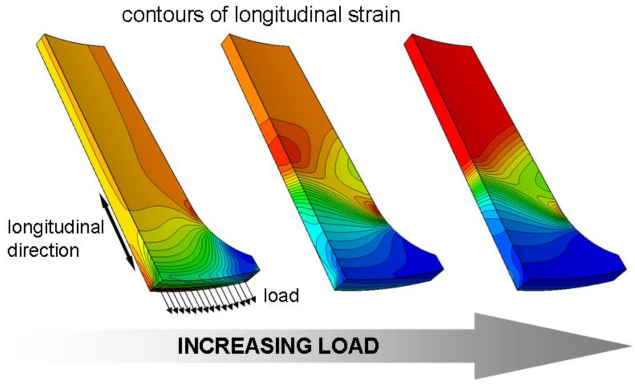

Figure 2: Finite element simulations provide full information in terms of space and time. This figure is an example result of an analysis of deformations in a curved wide plate specimen, performed at Laboratory Soete [22].

2.2 Degree of flexibility

Unavoidably, the amount of test material available in an experimental programme will be limited. Hence, it may be difficult to investigate the effect of variations of one specific geometrical or material parameter. Numerical simulations do not show this restriction, as every parameter can be freely adapted to the desires of the researcher. This paper elaborates two examples, which illustrate the flexibility of finite element simulations: firstly, the accurate description of a specific weld geometry, and secondly, the execution of a parametric study. Both studies have been performed at Laboratory Soete.

Firstly, it is possible to accurately describe any realistic weld geometry in an accurate way. To that purpose, a technique of node coordinate transformations has been developed. The principle is illustrated in Figure 3, where it is applied on an example of a weld (a). First, the paths of the fusion line and the weld cap are estimated (b). Next, these paths are described by mathematical equations (f_{bevel} and f_{cap}, respectively) (c). Using these equations, a coordinate transformation is then defined, starting from a perfectly regular rectangular grid (d). This transformation is finally applied on all nodes of a similar rectangular mesh (e). Using this technique, a good correspondence between the desired and the obtained mesh geometry is achieved (f). Moreover, the initially rectangular mesh is independent of weld geometry. Hence, a standard model can be built for all weld models, which is then transformed into the desired configuration. This allows for a high degree of automation.

Secondly, scripts that automate the entire process of performing finite element simulations, allow for an easy execution of parametric studies. In that way, the importance of, and the interaction between different parameters can be identified. An example of the procedure is illustrated in Figure 4. A series of parameter sets, each of which defines one model, is input to a Python script. This script communicates with the finite element software ABAQUS®, which builds and executes each model. The requested results are written to text files, which are then post-processed using a Matlab® script. This script produces standardised Microsoft® Excel® spreadsheets and HTML files, which can be viewed in any browser.

To illustrate the possibilities of the above technique, Figure 5 shows the results of a parametric study. For an Y-shaped weld, with the weld metal stronger than the connected base metals, the bevel angle 2α was varied between 0° and 80°. Clearly, the opening of a present crack (CMOD – Crack Mouth Opening Displacement) is influenced by this bevel angle. Apparently, if stronger than the base metals, a wider weld shields the crack from any remotely applied deformations. The extent of this influence is highly significant, as it eventually determines the failure mode of the welded connection. Indeed, for narrow welds (α = 0° and α = 10°) on the one hand, the crack causes the weld to collapse, resulting in a reduced deformation capacity. For wide welds (α = 20° and higher) on the other hand, the crack stabilizes. In the latter case, failure is no longer determined by the crack, but by the deformation capacity of the base metal. It should be remarked that possible stable crack extension has not been included in these analyses.

Copyright © 2010 by Labo Soete

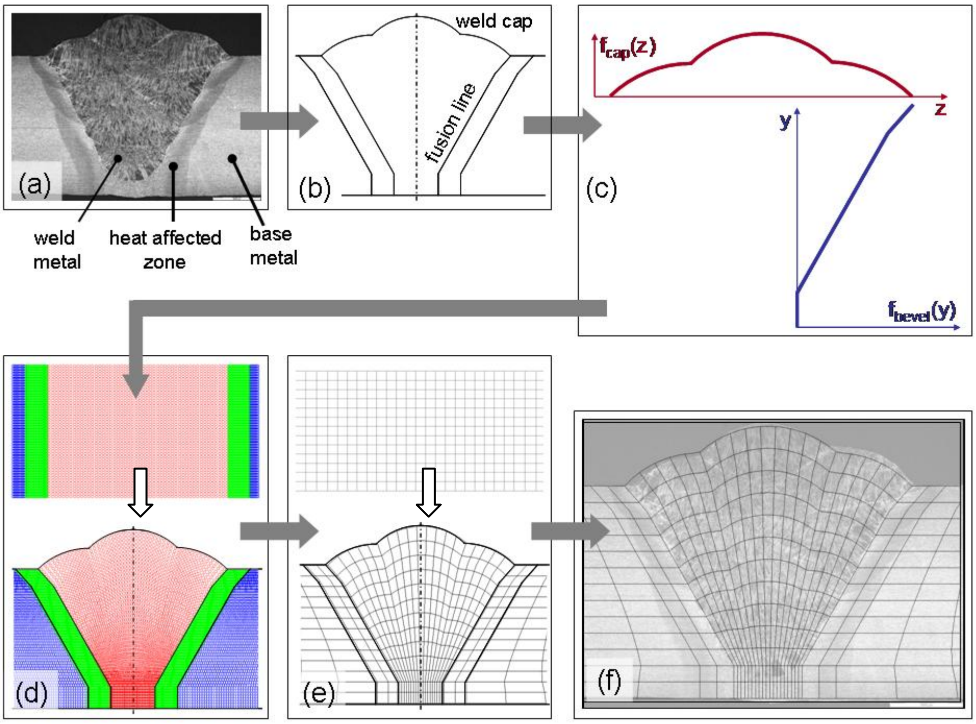

Figure 3: Coordinate transformations are used to create customized geometries.

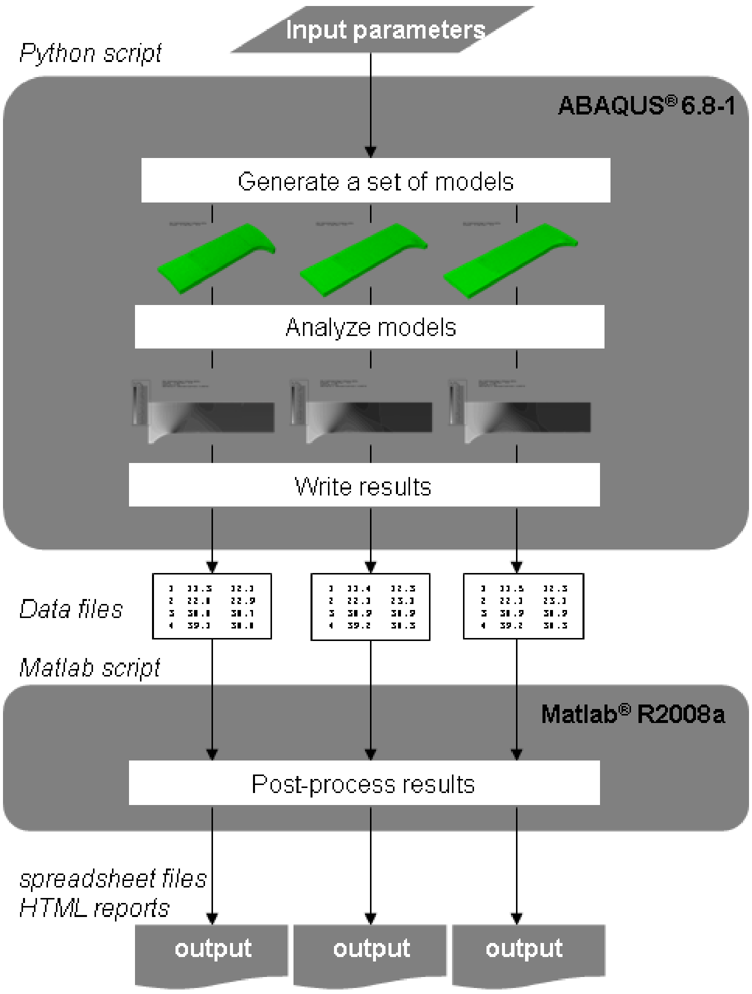

Figure 4: Structure of a parametric finite element study [22].

 Copyright © 2010 by Labo Soete

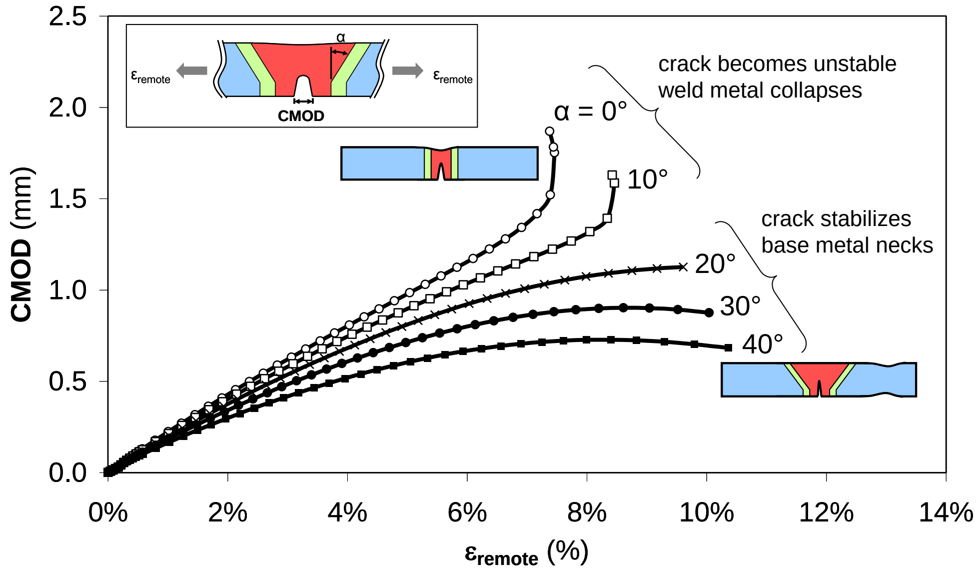

Figure 5: Example result of a parametric finite element study (red: weld, green: heat-affected zone, blue: base metal).

2.3 Degree of control

Despite the fact that an experiment is performed in a controlled environment, there will always be a certain degree of uncertainty. This may affect the correct interpretation and repeatability of a test result.

For instance, Figure 6 shows a tensile test specimen, containing a weld, which was intended to measure deformations. To that purpose, pins were spot-welded on the specimen, in order to mount LVDTs and clip gauges. However, Figure 6 clearly shows that some of the pins' angular positions were affected by the deformation of the specimen, resulting in a hardly interpretable measurement after the onset of necking.

Figure 6: A tensile test specimen, on which spot-welded pins were welded to mount LVDTs and clip gauges.

Finite element simulations do not show the problem of uncertainty of circumstances. Indeed, they are created in a numerical environment, where every aspect of the situation (geometry, material, load, boundary conditions) is, in principle, under complete control. This eliminates possible sources of a bad outcome and facilitates the comparison between different results.

3 LIMITATIONS OF FINITE ELEMENT MODELLING OF PIPELINE GIRTH WELDS

Despite providing certain advantages as compared to experiments, finite element models show some inherent limitations that cannot be ignored. These limitations are situated in the inability of perfectly describing the required input parameters, and the unavoidable occurrence of numerical errors. Each limitation is separately discussed in the sections below.

3.1 Limitations with respect to a realistic description of material behaviour

A numerical simulation is merely the production of a series of 'zeros and ones', which, if carefully designed, provides an approximation of reality. Despite the large number of computational tools available, some physical phenomena are too complicated to perfectly represent in the field of finite element modelling. In

Copyright © 2010 by Labo Soete

particular, two aspects are relevant for defected welds: heterogeneity of the materials, and crack initiation and growth.

Firstly, a weldment is a continous transition between different microstructures, all of which have resulted from a specific mechanical and thermal cycle during production and welding. Noteworthy are the occurrence of a heat-affected zone near the weld's fusion lines, the rich composition of coarse and fine-grained weld metal zones, due to subsequent weld passes (Figure 7; see also Figure 3(a)), but also the heterogeneity of the base metal (for instance, in the case of pipelines, due to pipeline forming).

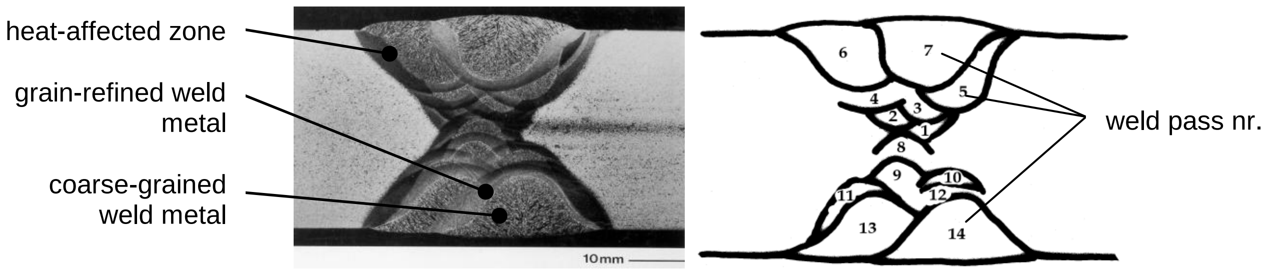

Figure 7: Heterogeneity in and around a weld metal, due to subsequent thermal cycles for each weld pass [1].

A heterogeneous material structure can be described in a finite element model. However, since it is extremely difficult or even impossible to accurately predict, or to experimentally retrieve the mechanical properties at every point in and around the weld, it is also impossible to describe the required input data for such heterogeneous structure in a numerical simulation. Approximations have to be made by using representative values, resulting from a number of characteristic tensile tests or simplified analytic predictions. Since the presence of heterogeneity has an unmistakable influence on the overall structural behaviour of the weldment [24-26], these approximations should be worst-case.

Secondly, the stress-strain concentration around a defect tip can give rise to the initiation and, subsequently, growth of a crack. The extent and direction of this crack growth depends on the applied loading, the degree of tri-axiality ('constraint') of the stress distribution at the crack tip, and the local microstructure. The first factor determines the crack driving force, whereas the last two factors define the so-called resistance curve. Expressed as a function of crack size, the amount of crack growth is determined by the intersection of the crack driving force curve with the resistance curve (Figure 8).

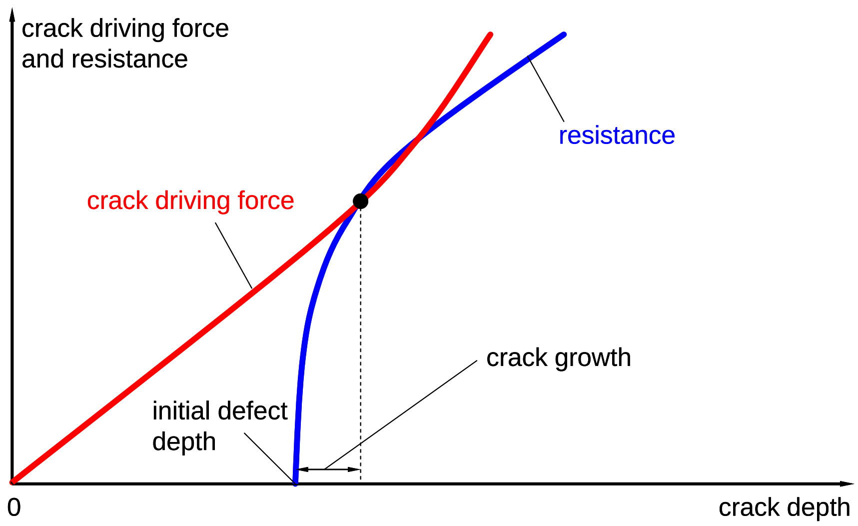

Figure 8: Crack growth is the result of a struggle between crack driving force and resistance.

Since the resistance curve depends on the local microstructure, which is highly heterogeneous, also the process of crack growth is characterized by a fair degree of scatter. To complicate matters even more, not only the extent, but also the direction of crack growth is determined by an interaction between applied

 Copyright © 2010 by Labo Soete

loading and local microstructure. This can give rise to fundamentally different failure mechanisms (Figure 9) [24;27].

Taking into account the abovementioned considerations and the fact that performing many simulations is time-consuming, finite element analyses should be restricted to the calculation of worst-case scenarios, under a set of conservative assumptions regarding material behaviour and crack growth properties. The results of these simulations then give a safe image of the qualitative influence of some parameters, more than an exact quantitative result. Alternatively, if the degree of scatter is known to some extent, numerical simulations can be used in the context of a probability-based design (stochastic finite element analysis). This, however, requires an extensive experimental characterization prior to simulating.

3.2 Numerical errors

Finite element modelling is a numerical technique, in which a continuous geometry is discretized into a finite set of elements. This discretization unavoidably involves the introduction of numerical errors. Choosing a larger set of smaller elements helps to reduce the order of magnitude of these errors, but requires more calculation time. Hence, choosing the mesh density is a trade-off between desired accuracy and available time. This trade-off should be chosen after a convergence study, taking into account the purpose of the analysis (Figure 10). If, on the one hand, the aim of the numerical model is to perform a parametric study based on a large amount of simulations, the calculation time should certainly be optimized. The corresponding reduction in result accuracy should, however, be kept limited. On the other hand, if the aim of the numerical model is a thorough, unique analysis of one specific situation, a highly accurate but time-consuming analysis may be chosen.

Figure 11 shows an example of a mesh convergence study, applied on the same model as in Figure 5. On the one hand, the coarsest mesh calculates very fast (147 seconds), but inaccurately. On the other hand, the finest mesh calculates very accurately, but is over ten times more time-consuming (1607 seconds). Given the fact that the model was intended for a large number of simulations, a trade-off was chosen to give a calculation time of about 10 minutes per simulation (600 seconds).

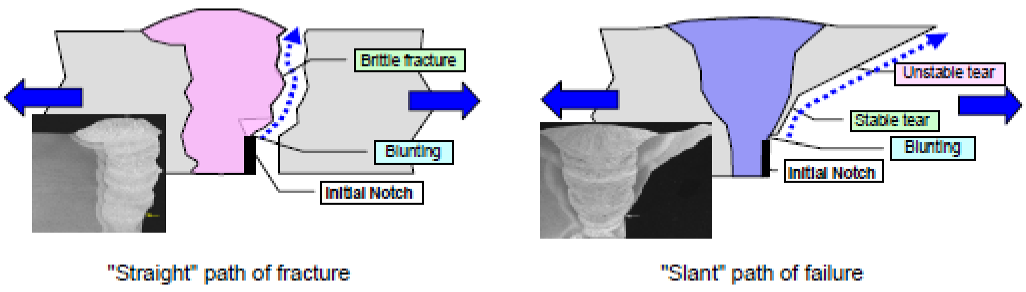

Figure 9: An initiated crack can grow in different directions, depending on the loading pattern and the local microstructures it encounters [24].

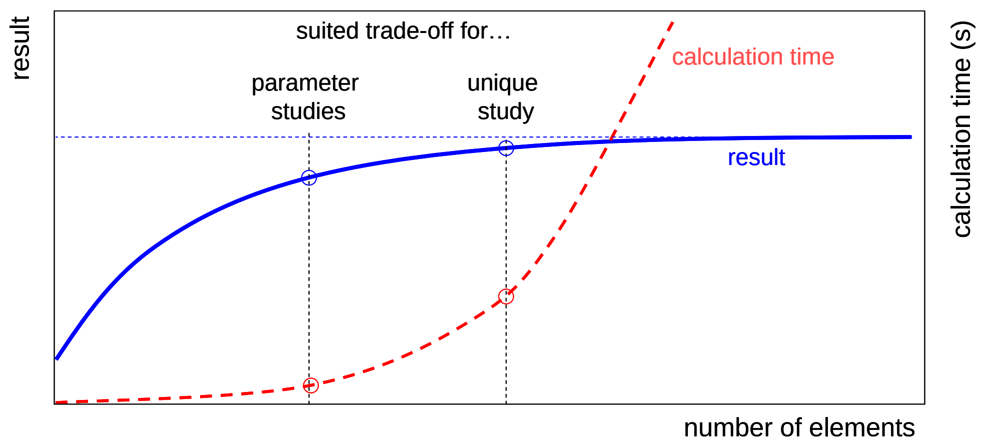

Figure 10: Philosophy behind a mesh convergence study.

Copyright © 2010 by Labo Soete

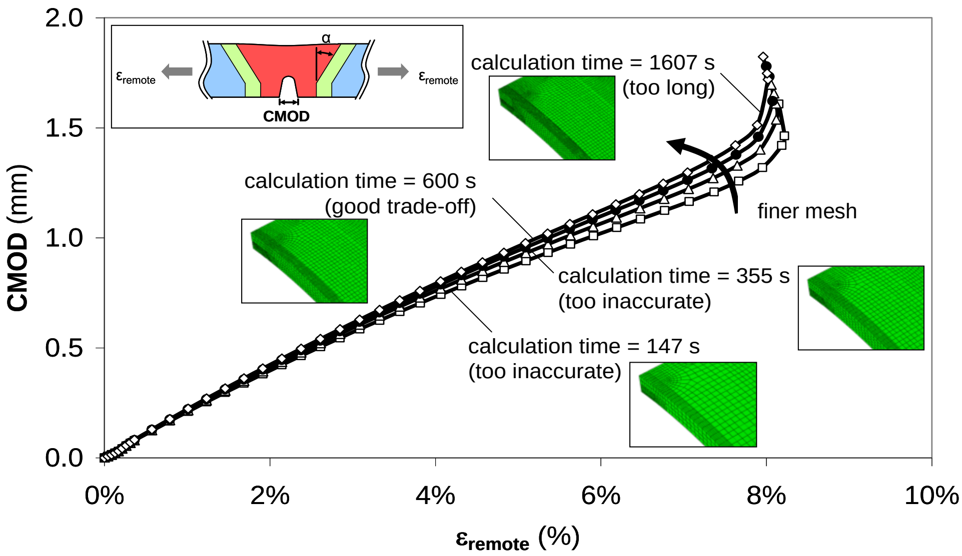

Figure 11: Example of a mesh convergence study.

4 CONCLUSIONS

Finite element simulations are powerful tools for the development of weld defect assessment procedures. They can provide an answer to the fact that not everything can be seen, changed, or avoided in an experiment. Indeed, a finite element simulation is under complete control of the user, and can be used to systematically investigate the influence of every single geometrical and material parameter.

However, the results of finite element simulations need to be interpreted with great care. A simulation is no more than a numerical attempt to describe the real situation through 'zeros and ones', and is therefore by definition an approximation. Moreover, every model should be consciously optimized in terms of accuracy and calculation time. The obtainable accuracy is highly dependent on the input parameters, which are not always deterministic. Hence, worst-case scenarios or probabilistic techniques should be considered.

Since approximations are unavoidable in a finite element simulation, the key to obtaining reliable results is a thorough experimental validation. Such validation can reveal the extent and acceptability of the idealizations made, and can allow for the postulation of conservative assumptions. That way, finite element simulations can be used to describe worst-case situations, resulting in a safe defect assessment.

5 ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of the FWO Vlaanderen (grant nrs. 1.1.880.09N and 1.5.247.08) and the IWT (grant nr. SB-091512).

6 REFERENCES

[1] A. Dhooge, *Metaalconstructies - Constructietechnieken*, 2010.
[2] E. Østby, E. Torselletti, E. Levold, *A strain-based approach to fracture assessment of pipelines*, Proceedings of FITNET, Amsterdam, The Netherlands, 2006.
[3] K. R. Jayadevan, E. Østby, C. Thaulow, *Fracture response of pipelines subjected to large plastic deformation under tension*, International Journal of Pressure Vessels and Piping, 81(9), 2004, 771-783.
[4] W. C. Mohr, *Pressure effects on strain concentration and constraint for strain-based design*, Proceedings of the 17th International Offshore and Polar Engineering Conference, 3121-3128, Lisbon, Portugal, July 2007.
[5] Y. Y. Wang, M. Liu, *The role of anisotropy, toughness transferability, and weld misalignment in the strain based design of pipelines*, Proceedings of the 17th International Offshore and Polar Engineering Conference, 3164-3171, Lisbon, Portugal, July 2007.
[6] E. Østby, C. Thaulow, Z. L. Zhang, *Numerical simulations of specimen size and mismatch effects in ductile crack growth - Part I: Tearing resistance and crack growth paths*, Engineering Fracture Mechanics, 74(11), 2007, 1770-1792.

Copyright © 2010 by Labo Soete

[7] E. Østby, C. Thaulow, Z. L. Zhang, *Numerical simulations of specimen size and mismatch effects in ductile crack growth - Part II: Near-tip stress fields*, Engineering Fracture Mechanics, 74(11), 2007, 1793-1809.

[8] D. Horsley, Y. Y. Wang, *Weld mismatch effects on the strain behaviour of flaws in X100 (Grade 690) pipe and girth welds*, Proceedings of the 4th International Conference on Pipeline Technology, 267-277, Ostend, Belgium, May 2004.

[9] C. Thaulow, Ø. Ranestad, M. Hauge, Z. L. Zhang, M. Toyoda, F. Minami, *FE calculations of stress fields from cracks located at the fusion line of weldments*, Engineering Fracture Mechanics, 57(6), 1997, 637-651.

[10] M. Liu, Y. Y. Wang, D. Horsley, *Significance of HAZ softening on strain concentration and crack driving force in pipeline girth welds*, Proceedings of the 24th International Conference on Offshore Mechanics and Arctic Engineering, Halkidiki, Greece, June 2005.

[11] Y. Y. Wang, M. Liu, D. Horsley, *Application of constraint-sensitive fracture mechanics to the assessment of girth weld integrity*, Proceedings of the 5th International Pipeline Conference, Calgary, Alberta, Canada, October 2004.

[12] D. M. Duan, Y. Y. Wang, Y. Chen, J. Zhou, *Modeling and CMOD mapping of surface-cracked wide plates*, Proceedings of the 7th International Pipeline Conference, Calgary, Alberta, Canada, October 2008.

[13] K. R. Jayadevan, E. Berg, C. Thaulow, E. Østby, B. Skallerud, *Numerical investigation of ductile tearing in surface cracked pipes using line-springs*, International Journal of Solids and Structures, 43(7-8), 2006, 2378-2397.

[14] E. Berg, B. Skallerud, C. Thaulow, K. Holthe, *Ductile fracture of pipelines - effects of constraint correction and circumferential crack growth*, Proceedings of the 17th International Offshore and Polar Engineering Conference, 3224-3230, Lisbon, Portugal, July 2007.

[15] K. Minnaar, P. C. Gioielli, M. L. Macia, F. Bardi, N. E. Biery, W. C. Kan, *Predictive FEA modeling of pressurized full-scale tests*, Proceedings of the 17th International Offshore and Polar Engineering Conference, 3114-3120, Lisbon, Portugal, July 2007.

[16] S. A. Kibey, K. Minnaar, J. A. Issa, P. C. Gioielli, *Effect of misalignment on the tensile strain capacity of welded pipelines*, Proceedings of the 18th International Offshore and Polar Engineering Conference, 90-95, Vancouver, BC, Canada, July 2008.

[17] S. A. Kibey, J. A. Issa, X. Wang, K. Minnaar, *A simplified, parametric equation for prediction of tensile strain capacity of welded pipelines*, Proceedings of the 5th International Conference on Pipeline Technology, Ostend, Belgium, October 2009.

[18] S. A. Kibey, K. Minnaar, W. Cheng, X. Wang, *Development of a physics-based approach for the prediction of strain capacity of welded pipelines*, Proceedings of the 19th International Offshore and Polar Engineering Conference, 132-137, Osaka, Japan, June 2009.

[19] X. Wang, S. T. Barbas, S. A. Kibey, P. C. Gioielli, K. Minnaar, *Validation of strain capacity prediction method: comparison of full-scale test results to predictions from tearing analysis based on FEA*, Proceedings of the 5th International Conference on Pipeline Technology, Ostend, Belgium, October 2009.

[20] Y. Y. Wang, D. Rudland, R. Denys, D. Horsley, *A preliminary strain-based design criterion for pipeline girth welds*, Proceedings of the 4th International Pipeline Conference, Calgary, Alberta, Canada, September-October 2002.

[21] Y. Y. Wang, W. Cheng, M. McLamb, D. Horsley, J. Zhou, A. G. Glover, *Tensile strain limits of girth welds with surface-breaking defects - Part 1: An analytical framework*, Proceedings of the 4th International Conference on Pipeline Technology, 235-249, Ostend, Belgium, May 2004.

[22] S. Hertelé, W. De Waele, R. Denys, J. Van Wittenberghe, M. Verstraete, *Investigation of pipe strain measurements in a curved wide plate specimen*, Accepted for the 8th International Pipeline Conference, Calgary, Alberta, Canada, September-October 2010.

[23] J. R. Rice, *A path independent integral and the approximate analysis of strain concentration by notches and cracks*, Journal of Applied Mechanics, 35, 1968, 379-386.

[24] R. Denys, *Weld metal strength mismatch: past, present and future*, Proceedings of the International Symposium to Celebrate Prof. Masao Toyoda's Retirement from Osaka University, 115-148, Osaka, Japan, June 2008

[25] R. Denys, P. De Baets, A. Lefevre, W. De Waele, *Significance of the tension test in relation to plastic pipeline design*, Proceedings of the Pipe Dreamer's Conference, 173-182, Pacifico Yokohama, Japan, November 2002.

[26] R. Denys, W. De Waele, A. Lefevre, *Effect of pipe and weld metal post-yield characteristics on plastic straining capacity of axially loaded pipelines*, Proceedings of the 5th International Pipeline Conference, Calgary, Alberta, Canada, October 2004.

[27] R. Denys, P. De Baets, A. Lefevre, W. De Waele, *Material tensile properties in relation to the failure behavior of girth welds subject to plastic longitudinal strains*, Proceedings of the Pipe Dreamer's Conference, 159-172, Pacifico Yokohama, Japan, November 2002.

 Copyright © 2010 by Labo Soete